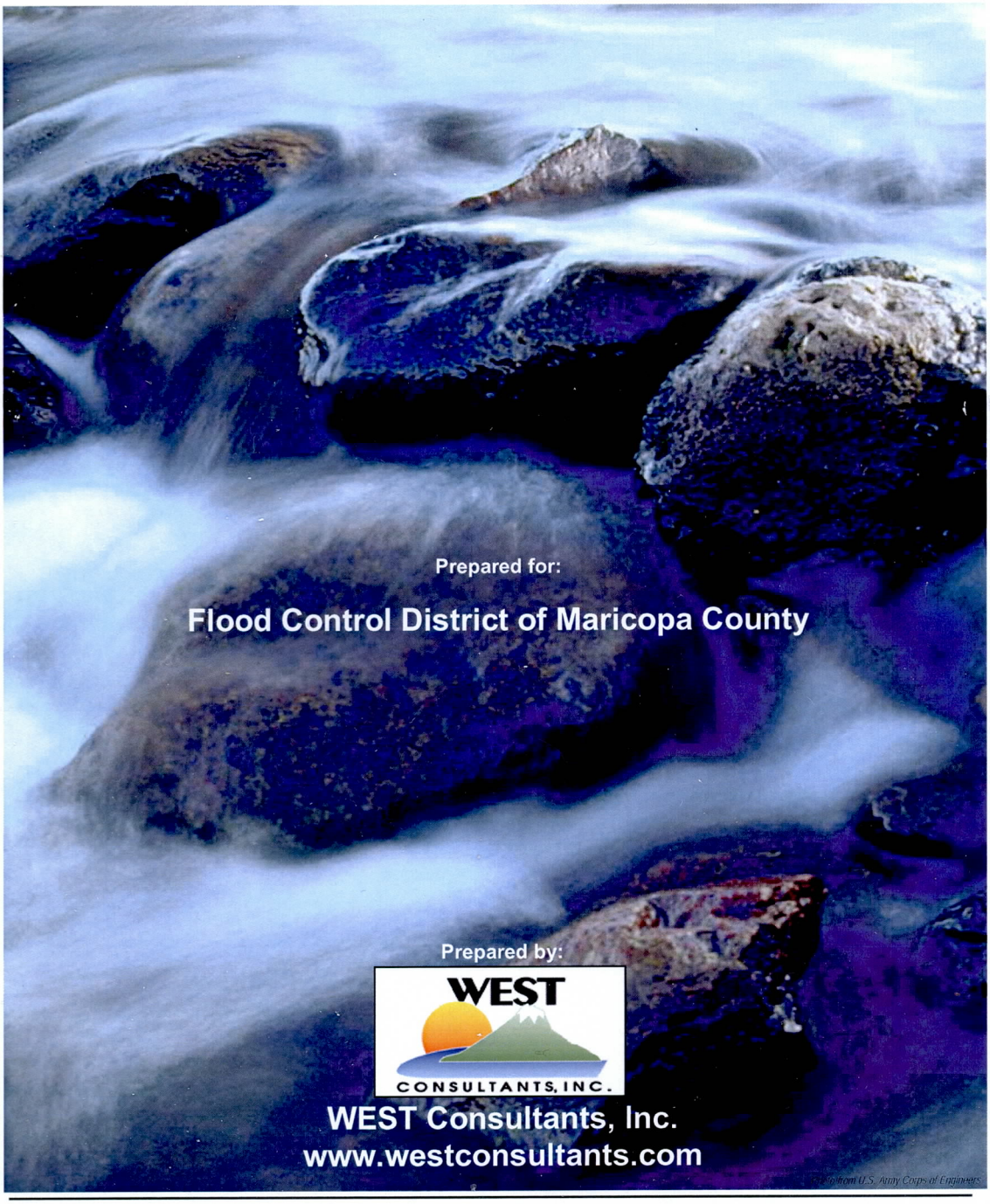
HEC-RAS Steady Flow and Unsteady Flow Short Course



Office Locations:

San Diego, CA
11440 W. Bernardo Court
Suite 360
92127-1644

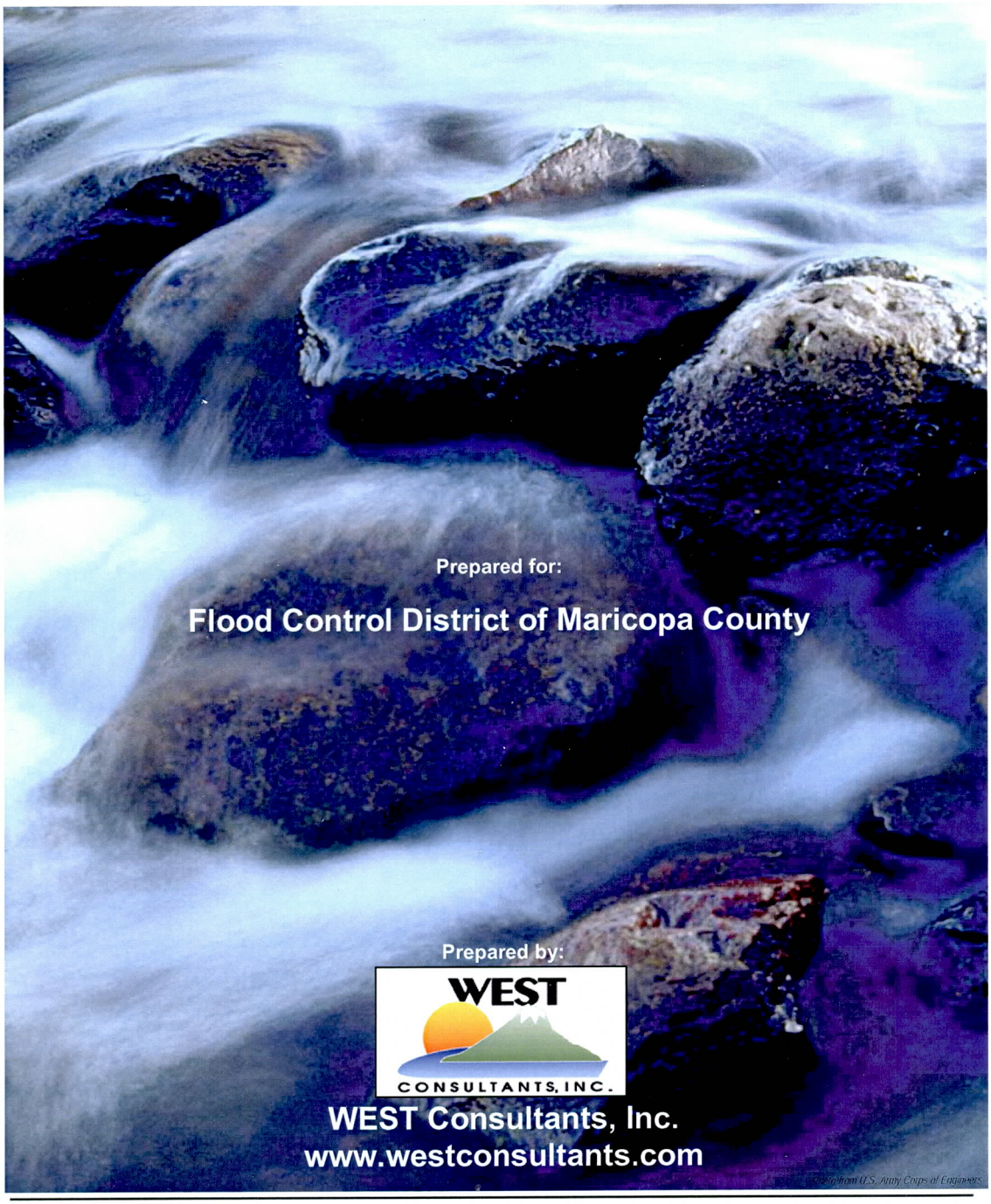
Bellevue, WA
12509 Bel-Red Road
Suite 100
98005-2535

Tempe, AZ
8950 S. 52nd Street
Suite 210
85284-1046

Salem, OR
2601 25th Street SE
Suite 450
97302-1286

1007.148

HEC-RAS Steady Flow and Unsteady Flow Short Course



Office Locations:

San Diego, CA
11440 W. Bernardo Court
Suite 360
92127-1644

Bellevue, WA
12509 Bel-Red Road
Suite 100
98005-2535

Tempe, AZ
8950 S. 52nd Street
Suite 210
85284-1046

Salem, OR
2601 25th Street SE
Suite 450
97302-1286

HEC-RAS, River Analysis System
Steady and Unsteady Flow
2-Day Course Outline

Presented by:

WEST Consultants, Inc.

Day 1

Event	Time
Introductions and Administration	8:00 AM - 8:15 AM
Lecture: Overview of HEC-RAS and Basic Data Requirements	8:15 AM - 9:15 AM
Break	9:15 AM - 9:30 AM
Workshop and Review: Developing a Hydraulic Model	9:30 AM - 10:30 AM
Workshop and Review: Adding Tributaries and Junctions	10:30 AM - 11:30 AM
Lunch	11:30 AM - 12:30 PM
Lecture: Channel Geometry and Modeling Techniques	12:30 PM - 1:30 PM
Workshop and Review: Creating Plans	1:30 PM - 2:45 PM
Break	2:45 PM - 3:00 PM
Lecture: Optional Capabilities of HEC-RAS	3:00 PM - 4:00 PM

Day 2

Event	Time
Lecture: Introduction to Unsteady Flow Modeling with RAS	8:00 AM - 9:00 AM
Break	9:00 AM - 9:15 AM
Lecture: Detailed Output and Results for Unsteady Flow Modeling	9:15 AM - 10:15 AM
Workshop and Review: Introduction to Unsteady Flow Modeling	10:15 AM - 11:15 AM
Lunch	11:15 AM - 12:15 PM
Lecture: Additional Geometric Data for RAS Unsteady Flow Models	12:15 PM - 1:15 PM
Lecture: Boundary and Initial Conditions in the RAS Unsteady Flow Data Editor	1:15 PM - 2:15 PM
Break	2:15 PM - 2:30 PM
Lecture: Model Stability, Accuracy, and Sensitivity	2:30 PM - 3:15 PM
Workshop and Review: Trouble Shooting	3:15 PM - 4:00 PM

WEST

CONSULTANTS, INC.
www.westconsultants.com

HEC-RAS, River Analysis System

By WEST Consultants, Inc.

Working with HEC-RAS: An Overview

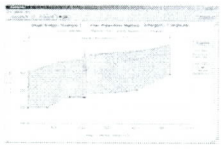
Note: Development of HEC-RAS is conducted at the U.S. Army Corps of Engineers Hydrologic Engineering Center in Davis, California.

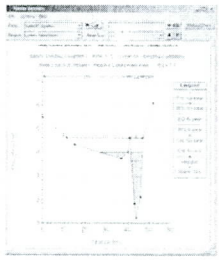
This document is property of WEST Consultants, Inc., and is not to be duplicated without prior consent

Introduction

WEST CONSULTANTS, INC.
www.westconsultants.com

- What is HEC-RAS?
  - An integrated package of hydraulic analysis programs
  - Uses Graphical User Interface (GUI)
- What can HEC-RAS do?
  - Steady flow water surface profiles
  - Unsteady flow water surface profile
  - Hydraulic design functions
    - Bridge scour
    - Uniform Flow
    - Stable channel design
    - Sediment transport capacity
    - Sediment Impact Assessment (SIAM)
  - Sediment analysis
  - Water quality (temperature) analysis





2

HEC-RAS is an integrated package of hydraulic analysis programs, in which the user interacts with the system through the use of a Graphical User Interface (GUI).

The system is capable of performing Steady and Unsteady Flow water surface profile calculations, and **will** include Sediment Transport, and several other hydraulic design computations in the future.

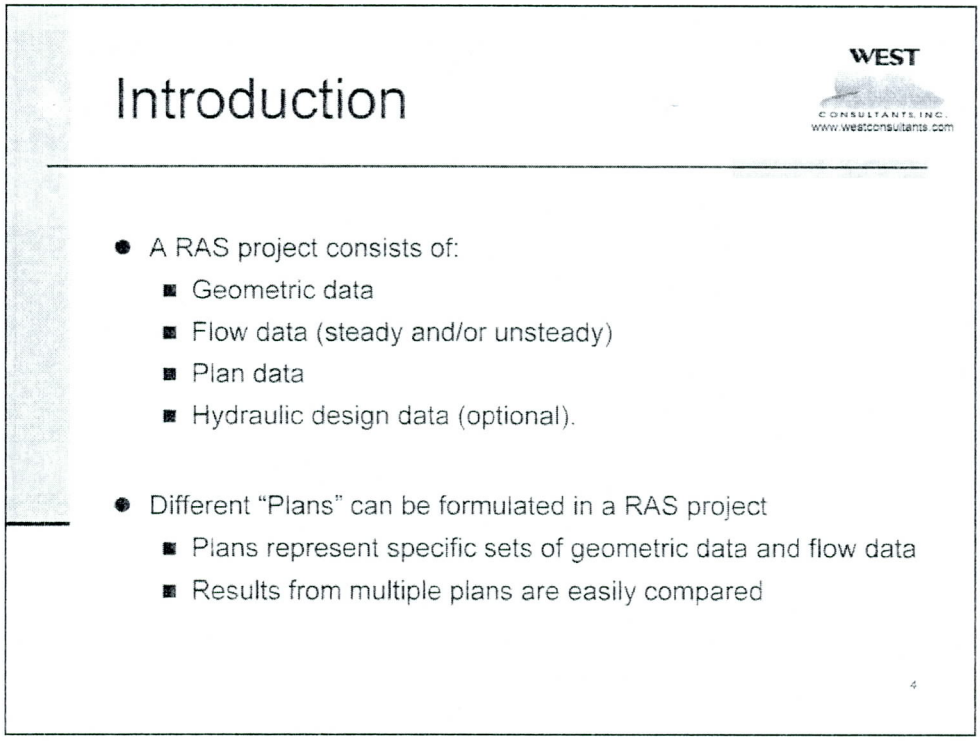
Introduction

WEST CONSULTANTS, INC.
www.westconsultants.com

- What is an HEC-RAS "Project"?
  - A set of data files associated with a particular river system
  - The User can perform various types of analyses within the HEC-RAS system

3

In HEC-RAS terminology, a Project is a set of data files associated with a particular river system. The modeler can perform any or all of the various types of analyses included in the HEC-RAS package as part of the project.



The data files for a project are categorized as follows: plan data, geometric data, steady flow data, unsteady flow data, and hydraulic design data.

During the course of a study, the modeler may want to formulate several different Plans.

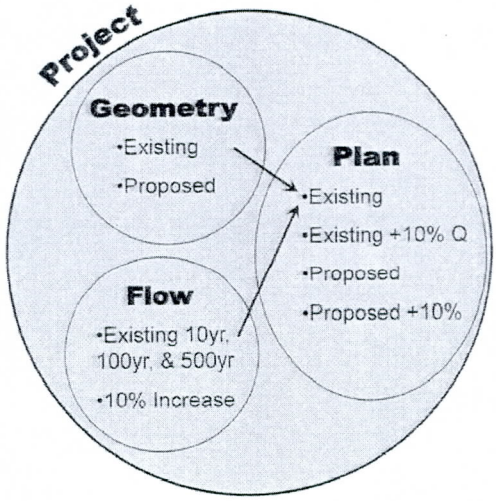
Each plan represents a specific set of geometric data and flow data. Once the basic data are entered into HEC-RAS, the modeler can easily formulate new plans.

New plans are easily formulated once the basic data are entered.

After simulations are made for the various plans, the results can be compared simultaneously in both tabular and graphical form.

RAS Project Schematic

WEST CONSULTANTS, INC.
www.westconsultants.com

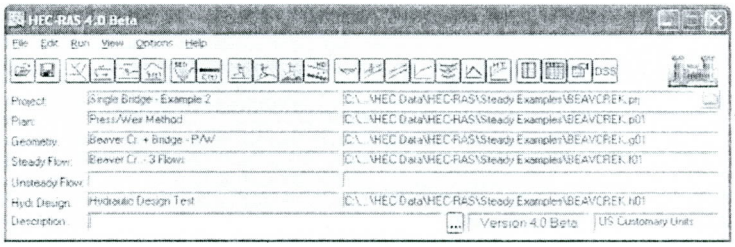


5

Starting HEC-RAS

WEST CONSULTANTS, INC.
www.westconsultants.com

- A new program group and program icon is created when you install the RAS program
- Double-click on the RAS icon from your Desktop to start the program



6

When you run the HEC-RAS Setup program, you automatically get a new program group and program icon for HEC-RAS in Windows.

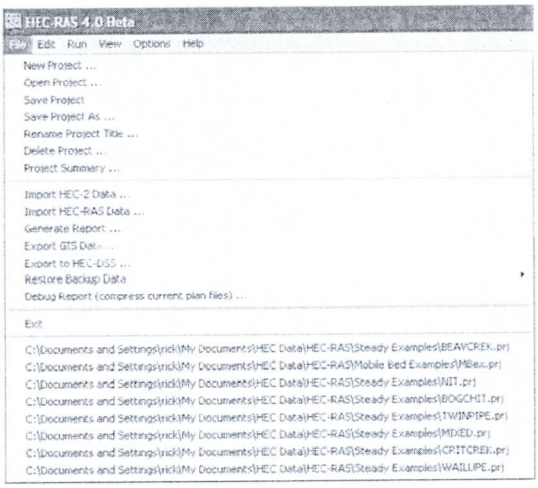
To start HEC-RAS from Windows, double-click on the HEC-RAS Icon.

When you first start HEC-RAS, you will see the main window as shown below (except you will not have any project files listed on your main window).

HEC-RAS Menu Bar

WEST CONSULTANTS, INC.
www.westconsultants.com

- File: Used for file management

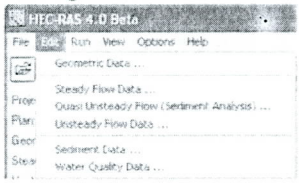


7

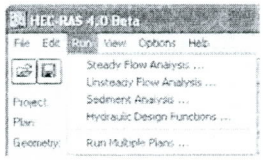
HEC-RAS Menu Bar

WEST CONSULTANTS, INC.
www.westconsultants.com

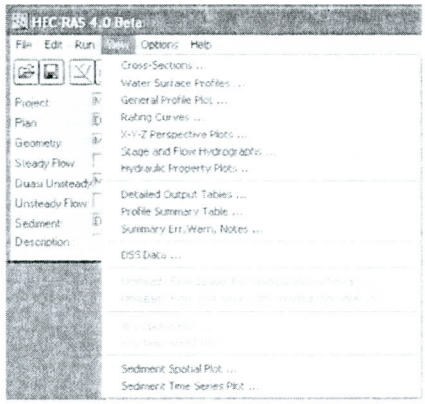
- Edit: Used for entering and editing data



- Run: Used to perform hydraulic calculations



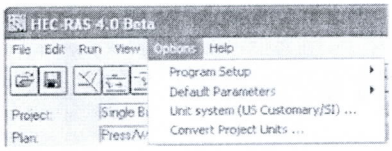
- View: Provides for graphical tabular displays of the model output



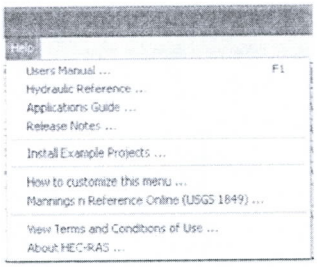
HEC-RAS Menu Bar

WEST CONSULTANTS, INC.
www.westconsultants.com

- Options: Program options



- Help: Access to the HEC-RAS manuals and pertinent information



9

Button Bar

WEST
CONSULTANTS, INC.
www.westconsultants.com

- On the HEC-RAS main window is a Button bar. The Button bar provides quick access to the most frequently used options under the HEC-RAS menu bar.

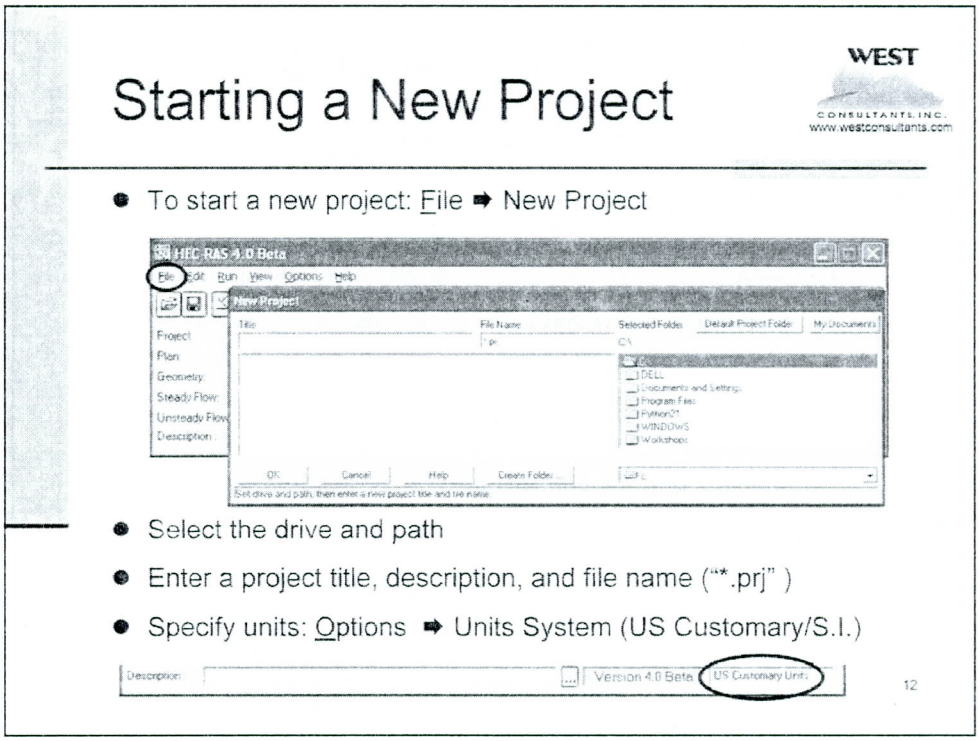
10

Steps in Developing a Hydraulic Model with HEC-RAS

WEST CONSULTANTS, INC.
www.westconsultants.com

- There are five main steps in creating a hydraulic model with HEC-RAS:
  1. Starting a new project
  2. Entering geometric data
  3. Entering flow data
  4. Performing the hydraulic calculations
  5. Viewing and printing results

11



To start a new project, go to the File menu on the main HEC-RAS window and select New Project, which brings up a New Project window.

You first select the drive and path that you want to work in (to actually select a path you must double click the directory you want in the directory box), then enter a project title and file name.

The project filename must have the extension ".prj" - the user is not allowed to change this.

After the OK button is pressed, a message box will appear with the title of the project and the directory where the project will be placed.

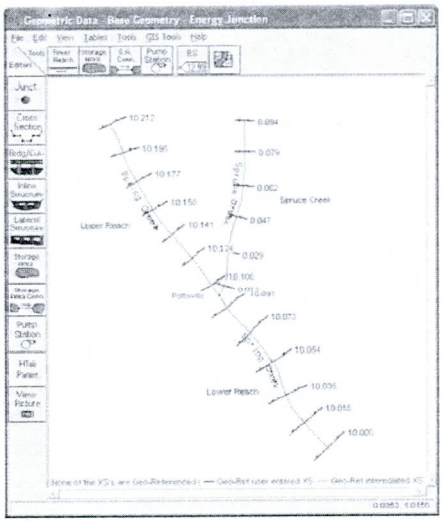
Note: Before any Geometric data and Flow data are entered, the user should select the Units System (English or Metric) that they would like to work in.

This is accomplished by selecting Unit System from the Options menu on the main HEC-RAS window.

Geometric Data Editor

WEST CONSULTANTS, INC. www.westconsultants.com

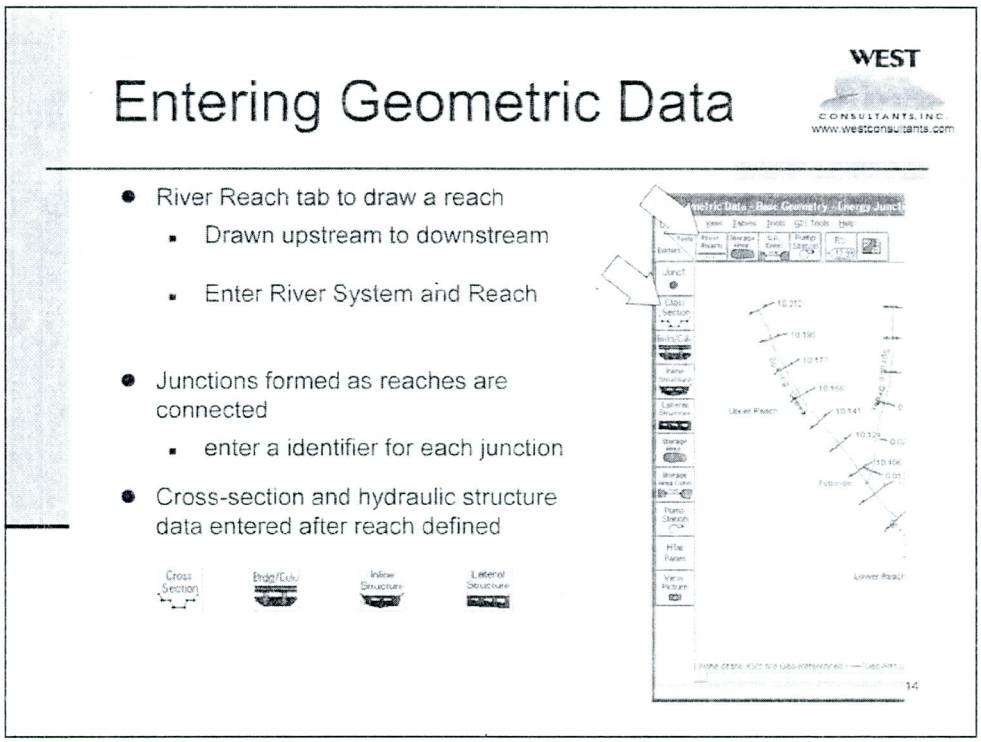
- Enter the necessary geometric data
  - Reaches, junctions (River System Schematic)
  - Cross-section data
  - Hydraulic structure data (bridges, culverts, weirs, etc.)
- Enter geometric data from the Edit menu



The next step is to enter the necessary geometric data, which consist of connectivity information for the stream system (River System Schematic), cross-section data, and hydraulic structure data (bridges, culverts, weirs, etc.).

Geometric data are entered by selecting Geometric Data from the Edit menu on the main HEC-RAS window.

Once this option is selected, the geometric data window will appear as show in the Figure (except yours will be blank when you first bring this screen up for a new project).

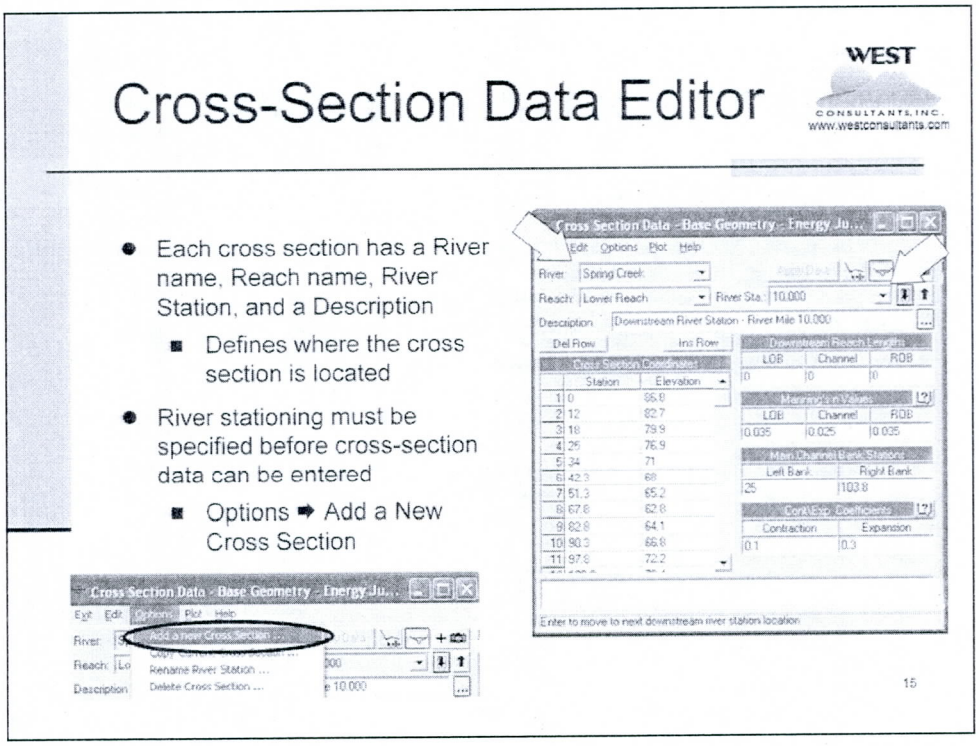


The geometric data is developed by pressing the River Reach button and drawing in a reach from upstream to downstream (positive flow direction).

After the reach is drawn, the user is prompted to enter a "River System" and a "Reach" identifier.

As reaches are connected together, junctions are automatically formed by the interface. The modeler is also prompted to enter an identifier for each junction.

After the river system schematic is drawn, the modeler can start entering cross-section and hydraulic structure data. Pressing the Cross Section button causes the cross section editor to pop up. This editor is shown in Figure 4.

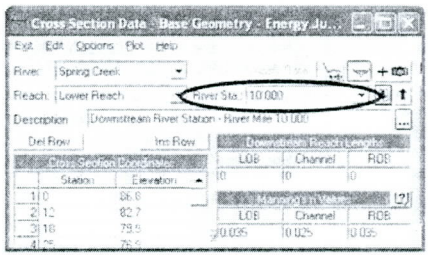


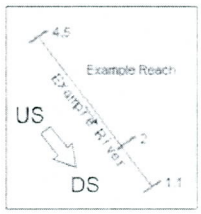
Each cross section has a River name, Reach name, River Station, and a Description. These identifiers are used to describe where the cross section is located in the river system and are input as shown in Figure 4.

Entering Cross-Section Data

WEST CONSULTANTS, INC. www.westconsultants.com

- The "River Station" must be a numeric value
  - Suggestion: corresponds to actual river station location (e.g., miles, kilometers)
- Cross sections ordered from highest river station upstream to the lowest river station downstream.





16

The "River Station" identifier does not have to be the actual river station (miles or kilometers) at which the cross section is located on the stream, but it does have to be a numeric value (e.g. 1.1, 2, 3.5, etc.).

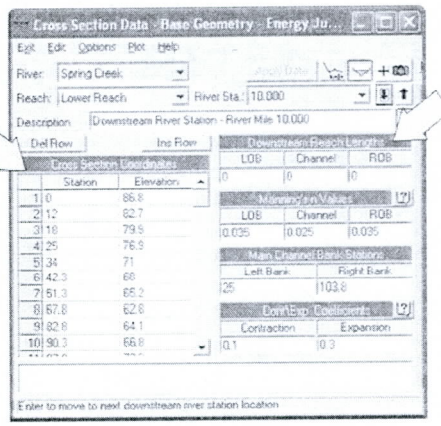
The numeric value is used to place cross sections in the appropriate order within a reach.

Cross sections are ordered within a reach from the highest river station upstream to the lowest river station downstream.

Cross-Section Data Editor

WEST CONSULTANTS, INC.
www.westconsultants.com

- Required data for each cross-section:
  - Station/Elevation data
  - Reach length
  - Manning's 'n' values
  - Main channel bank stationing
  - Contraction and expansion coefficients



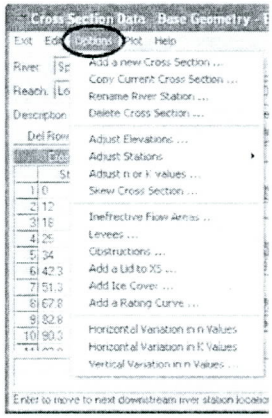
17

Each cross section has a River name, Reach name, River Station, and a Description. These identifiers are used to describe where the cross section is located in the river system and are input as shown in Figure 4.

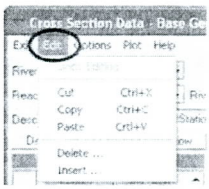
Cross-Section Data Options

WEST CONSULTANTS, INC. www.westconsultants.com

- Additional cross section features under Options



- Edit features are available to cut, copy, paste, insert, and delete data



18

Additional cross section features are available under Options from the menu bar. These options include:

adding, copying, renaming and deleting cross sections

adjusting cross section elevations, stations, and n or k-values

skewing cross section

ineffective flow areas; levees; blocked obstructions

adding a lid to a cross section

adding ice cover

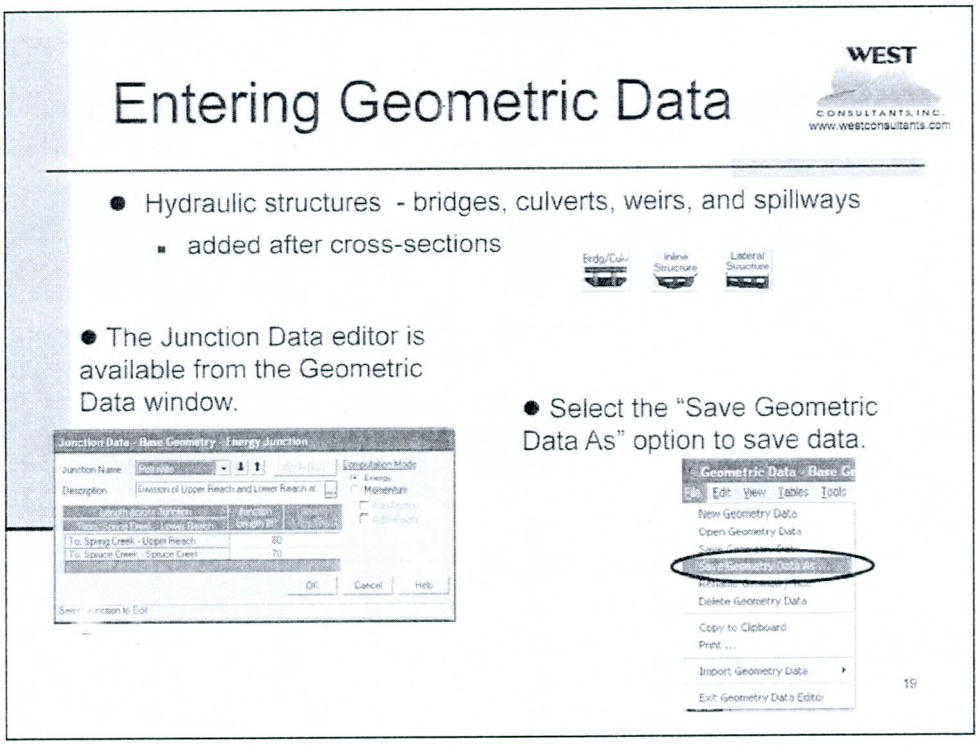
adding rating curve

horizontal variation of n or k-values

vertical variation of n-values

Also available from the Cross Section Data editor is the ability to plot any cross section or reach profile.

Edit features are available to cut, copy, paste, insert, and delete data from the Cross Section X-Y Coordinates grid.



Once the cross-section data are entered, the modeler can then add any hydraulic structures such as bridges, culverts, weirs and spillways.

Data editors, similar to the cross section data editor, are available for the various types of hydraulic structures.

If there are any stream junctions in the river system, additional data are required for each junction. The Junction Data editor is available from the Geometric Data window.

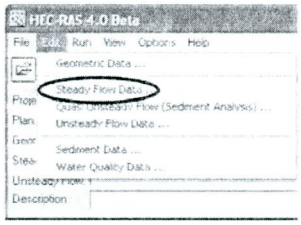
Once geometric data are entered, the data should be saved to a file on the hard disk. This is accomplished by selecting the "Save Geometric Data As" option from the File menu on the Geometric Data editor.

This option allows the user to enter a title for the geometric data. A filename is automatically established for the geometric data, and then saved to the disk. Once a title is established, geometric data can be saved periodically by selecting "Save Geometric Data" from the File menu of the Geometric Data editor.

Steady Flow Data Editor

WEST
CONSULTANTS, INC.
www.westconsultants.com

- Flow data should be entered after the geometry has been defined
- The Steady Flow Data Editor is available under the Edit menu bar option on the HEC-RAS main window.
- The button bar can also be used



20

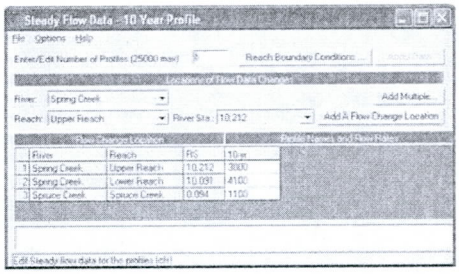
Once the geometric data are entered, the modeler can then enter any flow data that are required. The data entry form for flow data is available under the Edit menu bar option on the HEC-RAS main window.

An example of the flow data entry form is shown in Figure 5, which is the Steady Flow Data editor.

Entering Flow Data

WEST CONSULTANTS, INC.
www.westconsultants.com

- The Steady Flow Data consist of:
  - the number of profiles to be computed.
  - the flow data.
  - river system boundary conditions.
- At least one flow must be entered for every reach within the system.
  - flow can be changed at any location within the river system.
- Flow values must be entered for all profiles.
- Boundary conditions are required in order to perform the calculations.



21

The Steady Flow Data consist of: the number of profiles to be computed; the flow data; and the river system boundary conditions.

At least one flow must be entered for every reach within the system. Additionally, flow can be changed at any location within the river system. Flow values must be entered for all profiles.

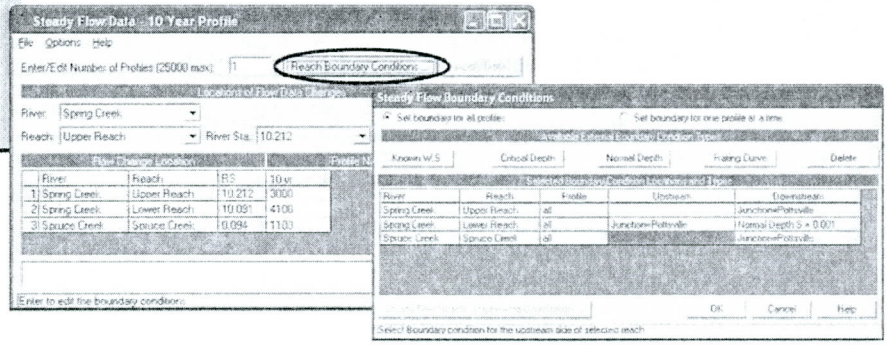
Boundary conditions are required in order to perform the calculations.

Entering Flow Data

WEST CONSULTANTS, INC. www.westconsultants.com

Boundary Condition Requirements

- Subcritical flow analysis - downstream boundary conditions
- Supercritical flow analysis - upstream boundary conditions
- Mixed flow regime calculation - both required



If a subcritical flow analysis is going to be performed, then only the downstream boundary conditions are required.

If a supercritical flow analysis is going to be performed, then only the upstream boundary conditions are required.

If the modeler is going to perform a mixed flow regime calculation, then both upstream and downstream boundary conditions are required.

The Boundary Conditions data entry form can be brought up by pressing the Reach Boundary Conditions button on the Steady Flow Data entry form.

Once all of the steady flow data and boundary conditions are entered, the modeler should save the data to the hard disk.

The user is only required to enter a title for the data, the filename is automatically assigned.

Performing the Hydraulic Computations

WEST CONSULTANTS, INC. www.westconsultants.com

- Once all of the geometric data and flow data are entered, the modeler can begin to perform the hydraulic calculations
- A plan is created with both a user specified geometry file and flow file
- Any of the available hydraulic design functions can be selected from the Run menu bar option on the HEC-RAS main window after a simulated has been run

23

Once all of the geometric data and flow data are entered, the modeler can begin to perform the hydraulic calculations.

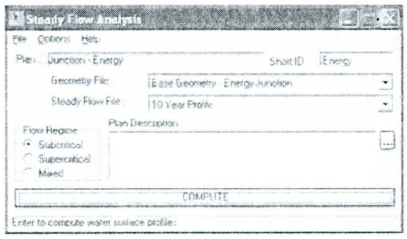
The modeler can also select any of the available hydraulic design functions from the Run menu bar option on the HEC-RAS main window.

An example of the simulation window is shown in Figure 6, which is the Steady Flow Analysis window.

Steady Flow Analysis Window

WEST CONSULTANTS, INC.
www.westconsultants.com

- To access the Steady Flow Analysis window:
  - Run ➡ Steady Flow Analysis
- A Plan can be put together by selecting "New Plan" from the File menu bar option of the Steady Flow Analysis window.
- Enter a Plan Title and Short Identifier (Short ID) and then specify a Geometry File and Steady Flow File
- The modeler can also select a Flow Regime:
  - Subcritical, Supercritical, or Mixed



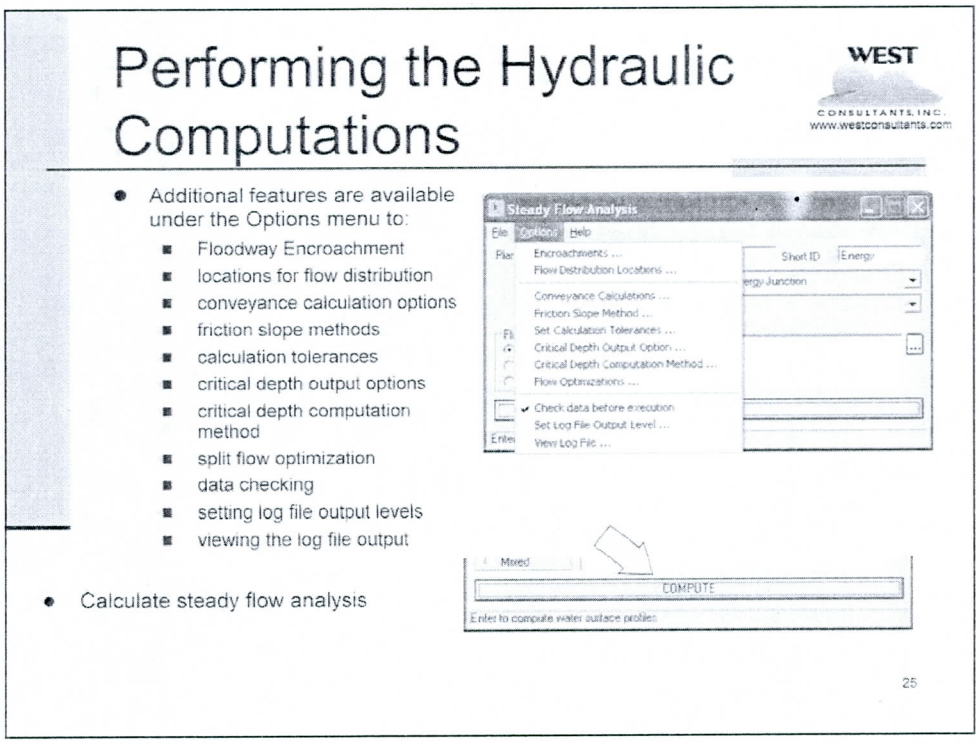
24

The modeler puts together a Plan by selecting a specific set of geometric data and flow data.

A Plan can be put together by selecting "New Plan" from the File menu bar option of the Steady Flow Analysis window.

Once a Plan Title and Short Identifier (Short ID) have been entered, the modeler can select a Flow Regime for which the model will perform calculations.

Subcritical, Supercritical, or Mixed flow regime calculations are available.



Additional features are available under the Options menu to:

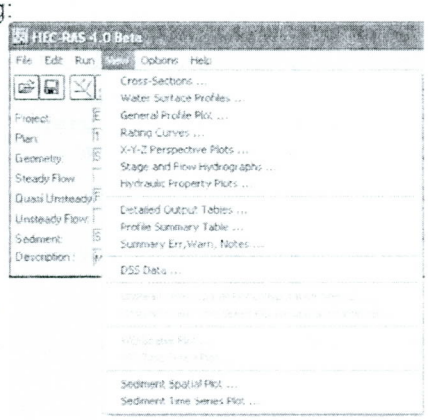
perform a Floodway Encroachment Analysis
set locations for calculating flow distribution output
conveyance calculation options
friction slope methods
calculation tolerances
critical depth output options
critical depth computation method
split flow optimization
data checking
setting log file output levels
viewing the log file output

The steady flow calculations can be performed by pressing the Compute button at the bottom of the Steady Flow Analysis window.

Viewing and Printing Results

WEST CONSULTANTS, INC.
www.westconsultants.com

- Several output features are available under the View option from the main window, including:
  - cross section plots
  - water surface profile plots
  - general profile plots
  - rating curve plots
  - X-Y-Z perspective plots
  - stage and flow hydrographs
  - hydraulic property plots
  - detailed output tables
  - profile summary table
  - summary of errors, warnings, and notes
  - Viewing data through DSS
  - Unsteady, water quality and Sediment plots



26

Several output features are available under the View option from the main window.

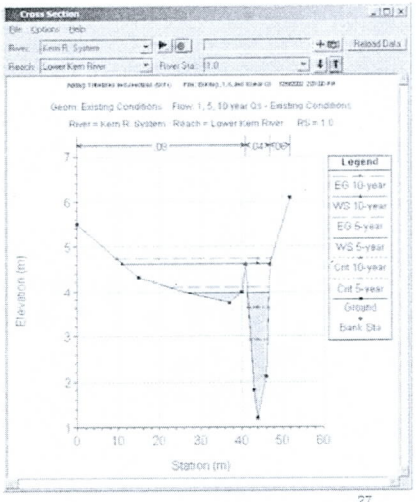
These options include:

cross section plots
water surface profile plots
general profile plots
rating curve plots
X-Y-Z perspective plots
stage and flow hydrographs
hydraulic property plots
detailed output tables
profile summary table
summary of errors, warnings, and notes
DSS data

Viewing and Printing Results

WEST CONSULTANTS, INC.
www.westconsultants.com

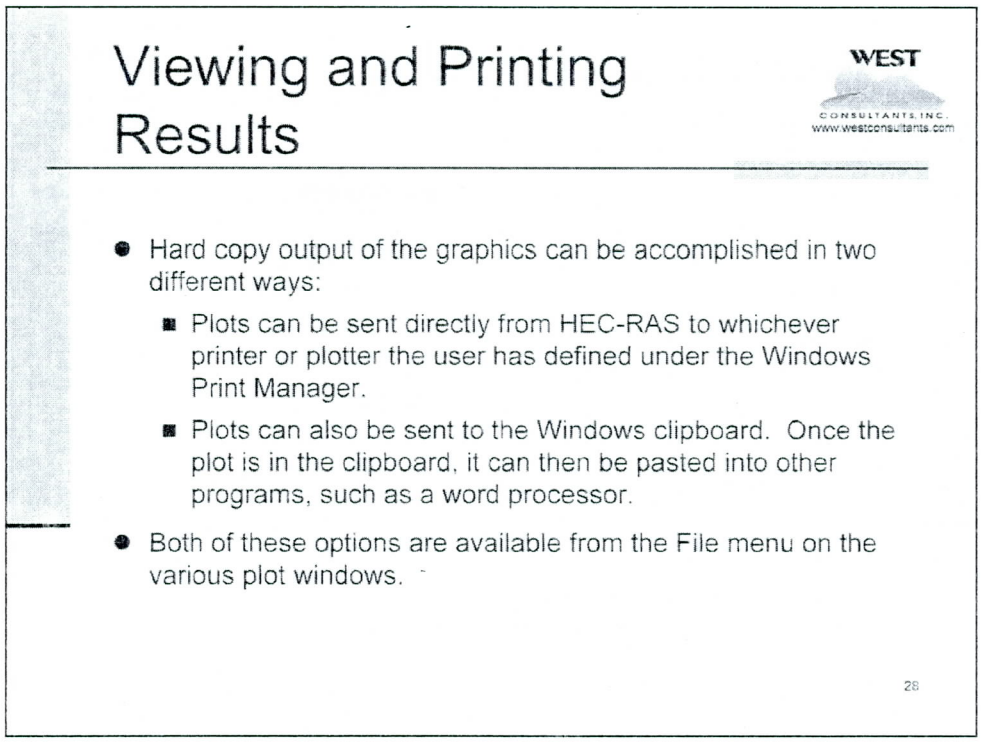
- Any cross-section can be viewed by selecting the River, Reach and River Station from the list boxes.
- Cross sections can be viewed by using the up and down arrow buttons or a drop-down list
- Several plotting features are available under the Options menu.
  - E.g., zoom in/out, change plans, modify line styles



27

An example of a cross section plot is shown in Figure 7. The user can plot any cross section by simply selecting the appropriate River, Reach and River Station from the list boxes at the top of the plot.

The user can also step through the plots by using the up and down arrow buttons. Several plotting features are available under the Options menu of the Cross Section plot.



Hard copy output of the graphics can be accomplished in two different ways:

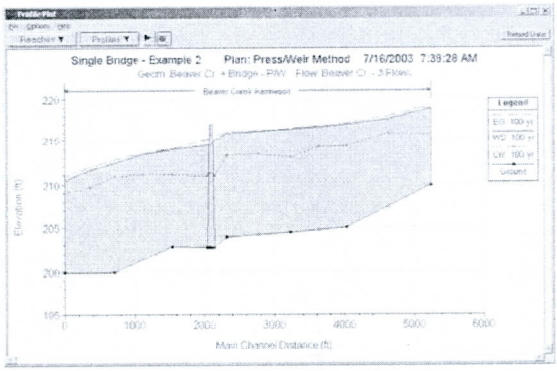
- Plots can be sent directly from HEC-RAS to whichever printer or plotter the user has defined under the Windows Print Manager.
- Plots can also be sent to the Windows clipboard. Once the plot is in the clipboard, it can then be pasted into other programs, such as a word processor.

Both of these options are available from the File menu on the various plot windows.

Viewing and Printing Results

WEST CONSULTANTS, INC.
www.westconsultants.com

- All of the options available in the cross section plot are also available in the profile plot.
- Additionally, the user can select which specific reaches to plot when a multiple-reach river system is being modeled.



29

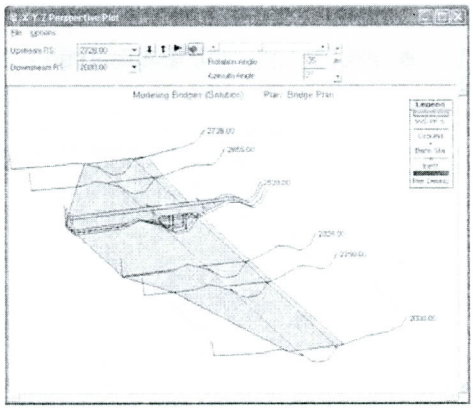
An example of a profile plot is shown in Figure 8. All of the options available in the cross section plot are also available in the profile plot.

Additionally, the user can select which specific reaches to plot when a multiple-reach river system is being modeled.

X-Y-Z Perspective Plot

WEST CONSULTANTS, INC.
www.westconsultants.com

- Plotting options:
  - Specifying starting and ending cross-sections
  - Rotation
  - Vary the horizontal to vertical ratio
- Multiple profiles can be overlaid on the cross section data.
- The graphic can be printed or sent through the Windows Clipboard to other programs.



30

The user has the option of defining the starting and ending location for the extent of the plot.

The plot can be rotated left or right, and up or down, in order to get different perspectives of the river reach.

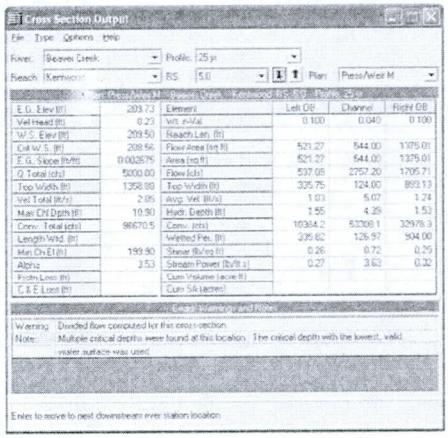
The computed water surface profiles can be overlaid on top of the cross section data.

The graphic can be sent to the printer or plotter directly, or the plot can be sent through the Windows Clipboard to other programs.

Detailed Tables

WEST CONSULTANTS, INC.
www.westconsultants.com

- Tabular output is available in two different formats:
  - "Detailed"
  - "Summary"
- Detailed tabular output provides detailed hydraulic results at a specific cross section location.



31

Tabular output is available in two different formats.

The first type of tabular output provides detailed hydraulic results at a specific cross section location (cross section table).

An example of this type of tabular output is shown in Figure 10.

Summary Tables

WEST CONSULTANTS, INC.
www.westconsultants.com

- "Summary Tables" show a limited number of hydraulic variables for several cross sections and multiple profiles.

Profile Output Table - Standard Table 1

File Options Std. Tables Locations Help

HEC-RAS Plan: Plan Avard Rivers: Bayou Dixie Reach: Kentwood

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 # Ch
			(cfs)	(ft)	(ft)	(ft)	(ft)	(ft/ft)	(ft/s)	(sq ft)	(ft)	
Kentwood	5.99	25 yr	5000.00	209.90	217.37	214.81	217.57	0.001952	5.10	2646.65	950.24	0.38
Kentwood	5.99	100 yr	10000.00	209.90	219.03	215.67	219.31	0.002019	6.56	4527.09	1801.43	0.43
Kentwood	5.99	May '74 flood	14000.00	209.90	219.99	217.38	220.30	0.002341	7.17	6597.56	1662.38	0.44
Kentwood	5.975*	25 yr	5000.00	207.45	215.99	214.61	216.38	0.002850	5.39	2172.03	1101.66	0.45
Kentwood	5.875*	100 yr	10000.00	207.45	217.96	215.90	218.23	0.001956	5.68	5247.37	1727.14	0.39
Kentwood	5.875*	May '74 flood	14000.00	207.45	218.98	216.91	219.25	0.001859	6.05	7044.58	1796.81	0.39
Kentwood	5.76	25 yr	5000.00	205.00	215.20	213.34	215.31	0.001167	3.45	3784.91	1490.56	0.29
Kentwood	5.76	100 yr	10000.00	205.00	217.46	214.30	217.57	0.000802	3.75	7373.12	1671.81	0.25
Kentwood	5.76	May '74 flood	14000.00	205.00	218.44	214.49	218.59	0.000900	4.34	9060.58	1763.64	0.27

Total flow in cross section.

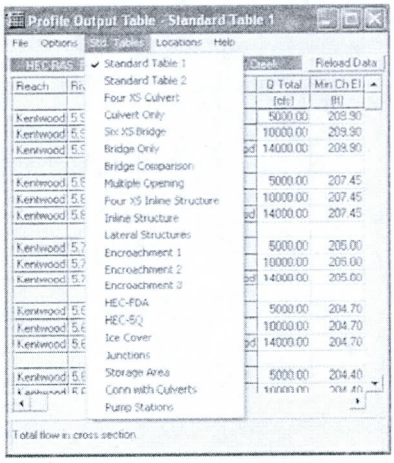
32

The second type of tabular output shows a limited number of hydraulic variables for several cross sections and multiple profiles. An example of this type of tabular output is shown in Figure 11.

Viewing and Printing Results

WEST CONSULTANTS INC.
www.westconsultants.com

- Several pre-defined summary tables are available
- Tabular output can be printed or passed through the clipboard.
  - Available under the File menu on each of the table forms.



33

There are several standard tables that are pre-defined and provided to the user under the Tables menu from the profile output tables.

Users can also define their own tables by specifying what variables they would like to have in a table.

User specified table headings can be saved and then selected later as one of the standard tables available to the project.

Tabular output can be sent directly to the printer or passed through the clipboard in the same manner as the graphical output described previously. This option is also available under the File menu on each of the table forms.

User-Defined Summary Tables

WEST CONSULTANTS, INC.
www.westconsultants.com

- Users can also define their own "summary" tables by specifying what variables they would like to have in a table.
- User specified table headings can be saved and then selected later as one of the standard tables available to the project.

34

WEST

CONSULTANTS, INC.
www.westconsultants.com

Channel Geometry and Modeling Techniques

- **Objective**: This lecture provides guidance on important geometric considerations associated with 1-D modeling

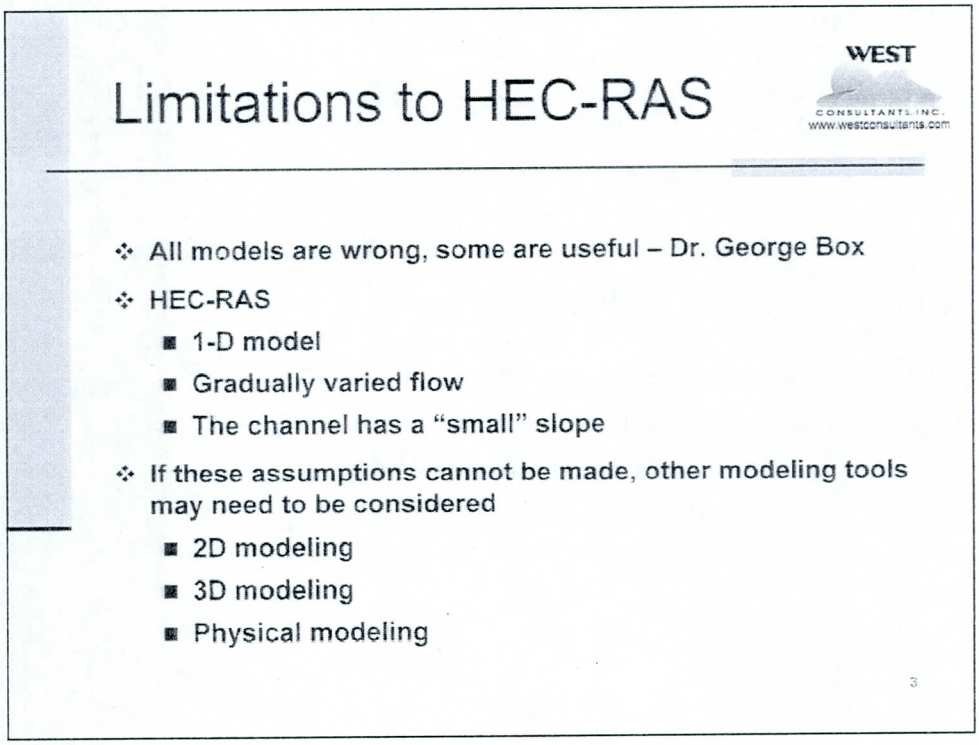
Note: Development of HEC-RAS and HEC-HMS is conducted at the U.S. Army Corps of Engineers Hydrologic Engineering Center in Davis, California

Objectives

WEST CONSULTANTS, INC.
www.westconsultants.com

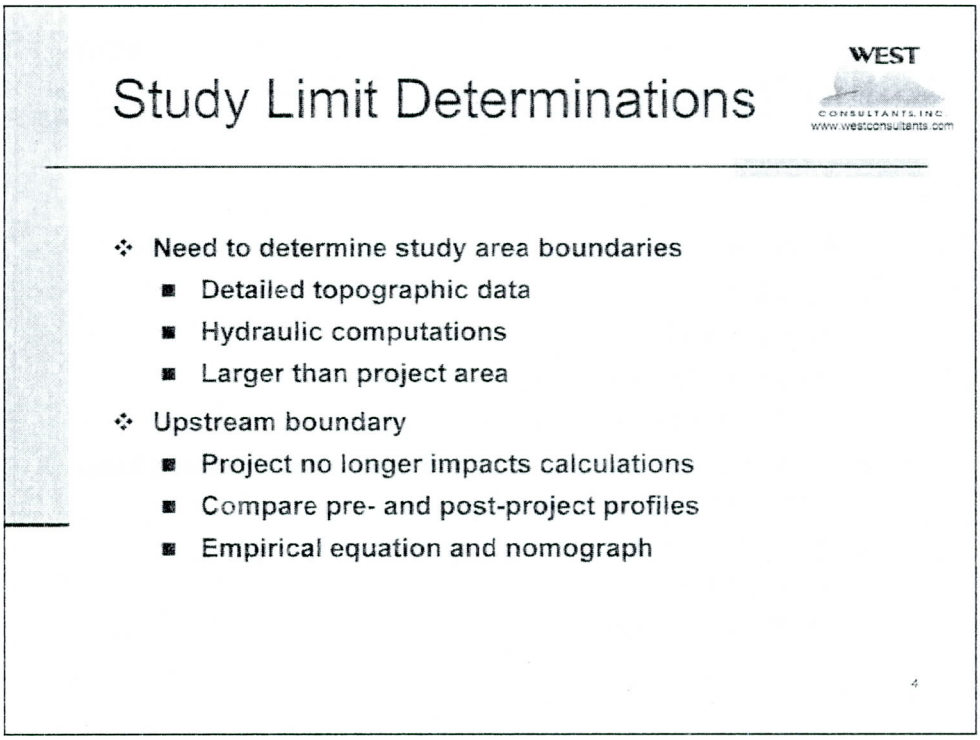
- Limitations to HEC-RAS
- Study limit determinations
- Geometric data
- Discharge data
- Roughness data
- Calibration
- Consistency checks

2



One-dimensional flow implies that depth and velocity vary only in the longitudinal direction of the channel (i.e., the energy grade line elevation and water surface elevation are constant across a given cross-section). Although the 1-D assumption is not theoretically correct, it is suitable for most open channel hydraulic work. The gradually varied flow assumption implies that hydrostatic pressure prevails and that vertical accelerations can be ignored. The exception to the gradually varied assumption occurs at structures such as bridges, culverts, inline structures, and lateral structures. At these locations, where the flow can be rapidly varied, the momentum equation or other empirical equations are used. Typically, the channel slope is considered "small" if it is less than 1:10. This is not a strict rule. For example, small, isolated reaches greater than 10% may work just fine. This breakpoint is simply where the depth measured parallel to gravity and the depth measured normal to the bed slope begin to diverge significantly.

The cost for developing 2-D and 3-D models are significantly higher than the cost associated with developing a 1-D model.

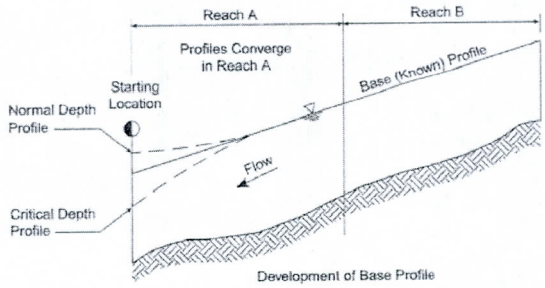


Determining the study limits defines the area where detailed topographic data are needed as well as where hydraulic computations are to be performed. The boundaries usually extend further upstream and downstream of the area of interest. The upstream boundary is usually a point where the impacts of the project (i.e., backwater effects) are no longer observed and hydraulic computations are no longer needed. To determine this, a pre- and post-project profile can be determined. Where these two profiles merge would be the appropriate location of the upstream boundary condition. HEC's *Accuracy of Computed Water Surface Profiles* (1986) presents an empirical equation and nomograph for locating the upstream boundary under subcritical flow as a function of hydraulic depth, head loss across the structure, and channel slope.

Study Limit Determinations

WEST CONSULTANTS, INC. www.westconsultants.com

- Downstream boundary
  - Eliminate errors in downstream boundary condition
  - Trial-and-error
  - Empirical equations and nomographs



Development of Base Profile

U.S. Army Corps of Engineers

5

The downstream boundary should be located sufficiently far enough downstream so that any errors associated with the boundary condition are damped out before the project area is reached. The location of the downstream boundary can be determined by a trial-and-error procedure. Also, HEC's *Accuracy of Computed Water Surface Profiles* (1986) presents empirical equations and nomographs for locating the downstream boundary as a function of hydraulic depth and channel slope.

Geometric Data

WEST CONSULTANTS, INC.
www.westconsultants.com

- Aerial photographs and site visits
- Flow visualization
- Bank station locations
- Reach lengths
- Ineffective flow areas
- Cross-section spacing/alignment

6

Aerial Photographs and Site Visits

WEST CONSULTANTS, INC. www.westconsultants.com

- Useful first steps
- Assists in:
  - Estimating roughness values
  - Placing cross-sections
  - Determining bank station locations
  - Visualizing flow patterns through/around structures
- Sources
  - Google Earth
  - Government and local agencies/websites
  - USGS quad sheets

7

Many websites have aerial photographs that can be viewed for free like the local county's assessor site or flood control district site. Google Earth is also a powerful tool for finding aerial photographs. Local universities may have map/photo collections for the state that they are located in. Many government agencies provide current and historical aerial photographs. A partial list of these agencies includes:

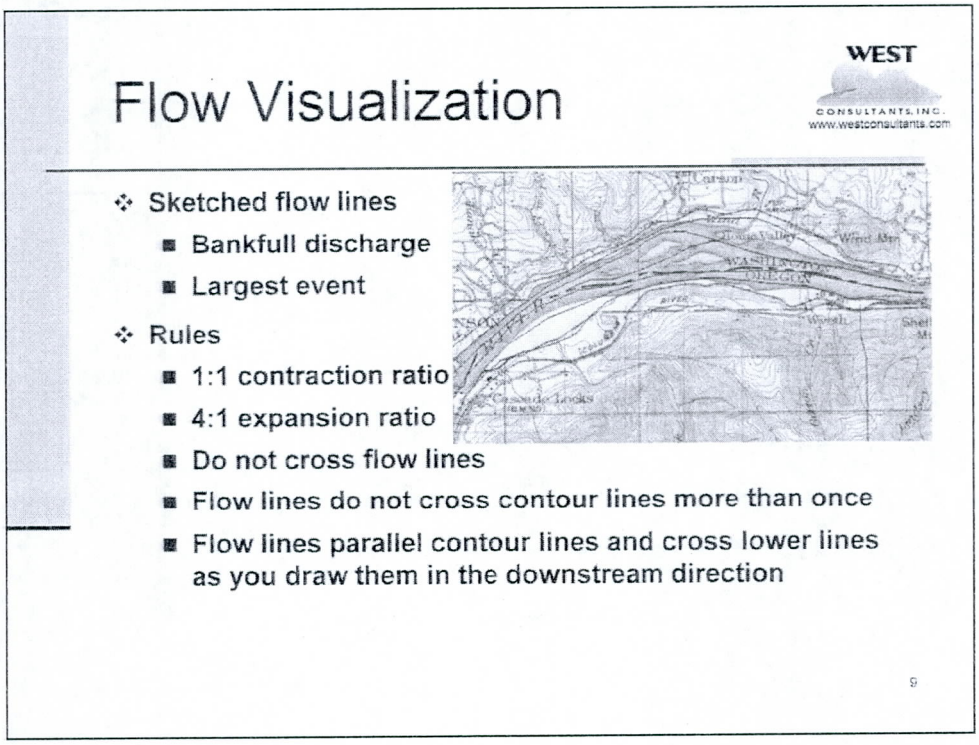
1. US Geological Survey http://edcsns17.cr.usgs.gov/finder/finder_main.pl?dataset_name=NAPP
2. Bureau of Land Management http://www.blm.gov/nstc/aerial/index.html
3. US Department of Agriculture (Farm Services Bureau) http://www.fsa.usda.gov/FSA/apfoapp?area=apfohome&subject=landing&topic=landing

Flow Visualization

WEST CONSULTANTS, INC.
www.westconsultants.com

- Identifying flow lines helps in model development
- Flow lines help identify
  - Ineffective flow areas
  - Areas of storage
  - Cross-section spacing and alignment

8



Quad sheets should be overlaid with mylar or other transparent film and flow lines sketched for bankfull and flood discharges (or largest event to be analyzed). If there are radical changes in flow area between these two discharges, you may want to add flow lines from an intermediate discharge.

Some general rules of thumb about drawing flow lines are as follows:

Remember to restrict the flow lines to a 1 to 1 contraction ratio and 4 or 3 to 1 expansion ratio.

Flow lines do not cross.

Flow lines do not generally cross contour lines more than once.

In a typical stream, flow lines generally parallel the contour lines and cross lower lines as you draw them in the downstream direction

Draw flow lines starting from the upstream end unless here are significant protruding land features – then see below.

Start drawing flow lines at the anticipated waters edge at protrusions into the water flow. Then draw upstream at the 1 on 1 ratio and downstream at the 1 on 3 or 4 ratio. Continue reasonably drawing these lines upstream and downstream until they hit the waters edge.

Bank Station Locations

WEST CONSULTANTS, INC.
www.westconsultants.com

- Difficulties
  - Delineating channel from overbanks
  - Competing goals
- Tools
  - Site visit
  - Aerial photographs
- Criteria
  - Vegetation
  - Soils
  - Topography

10

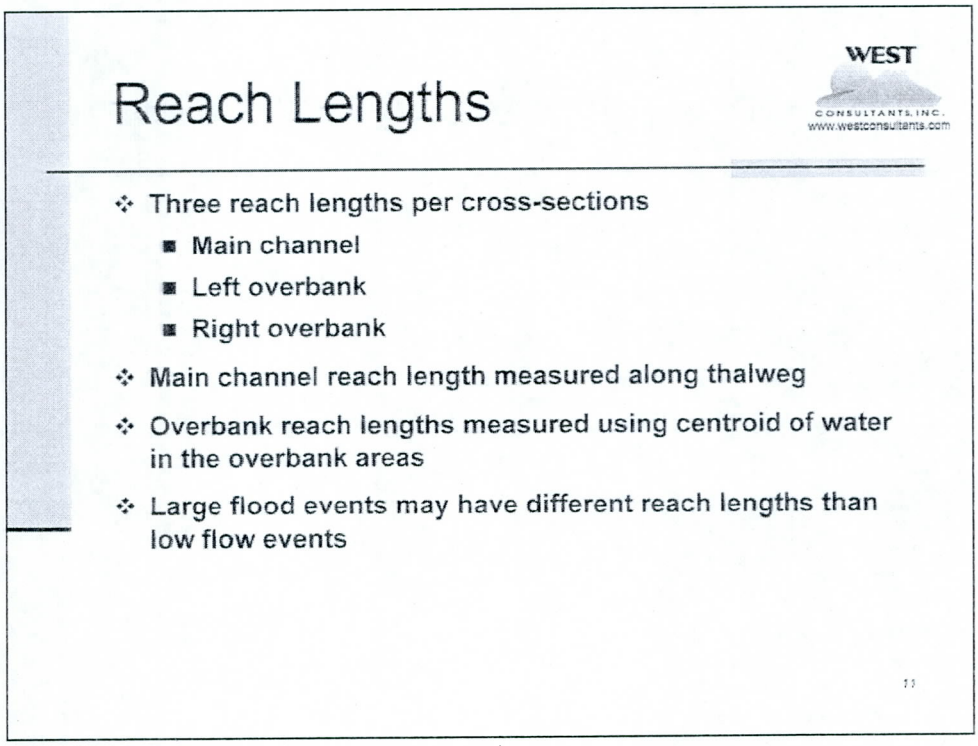
At times, it can be difficult to determine the channel banks are located. Sometimes, it may be difficult to delineate where the channel ends and where the overbanks begin. Also, the entities involved in the project may have competing goals that make it difficult to determine the bank station locations. A developer may want to place the bank stations in towards the channel so that there is more room to encroach. Local flood control agencies may want to push the banks stations away from the channel to be more conservative and provide better flood protection.

Site visits and aerial photographs are two important tools that the modeler can use to determine the bank station locations. Vegetation, soils, and topography are also valuable tools.

Vegetation – Bank stations are usually located at the point where vegetation along the stream corridor changes from terrestrial to aquatic species, or the point where permanent, terrestrial vegetation begins.

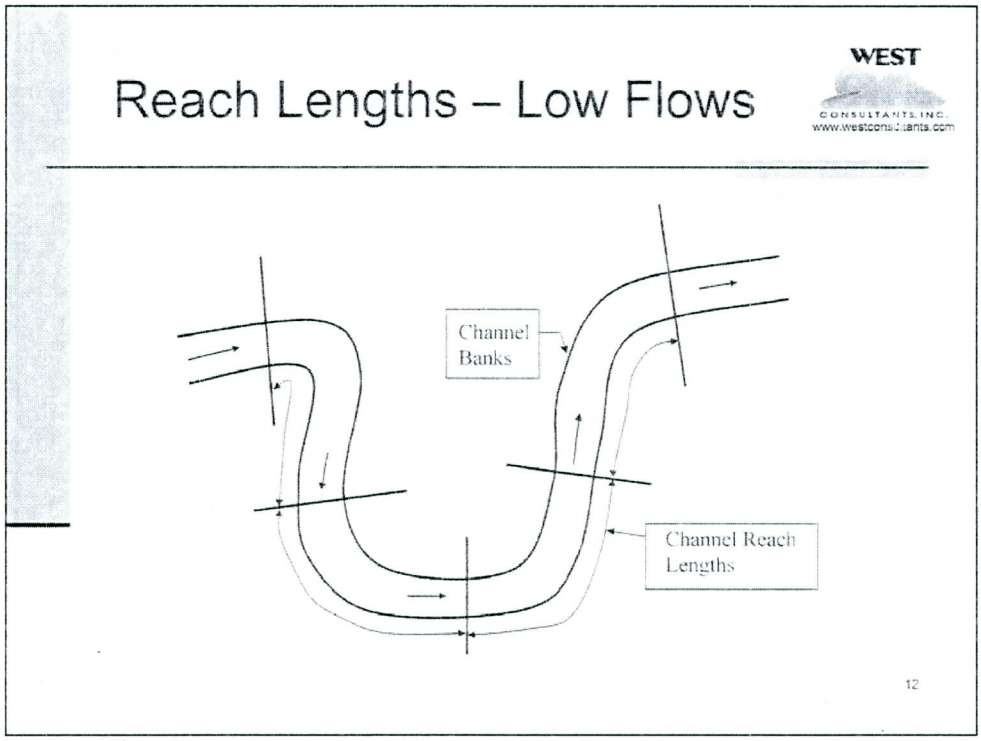
Soils – Banks stations are usually located at the point where soil characteristics change from undifferentiated, poorly developed, layered, fluvial deposits subject to scour and deposition to more well-developed soils with distinct soil.

Topography – Bank stations are usually located at a break in slope or at the point where the top of the channel bank transitions to the more planar floodplain.

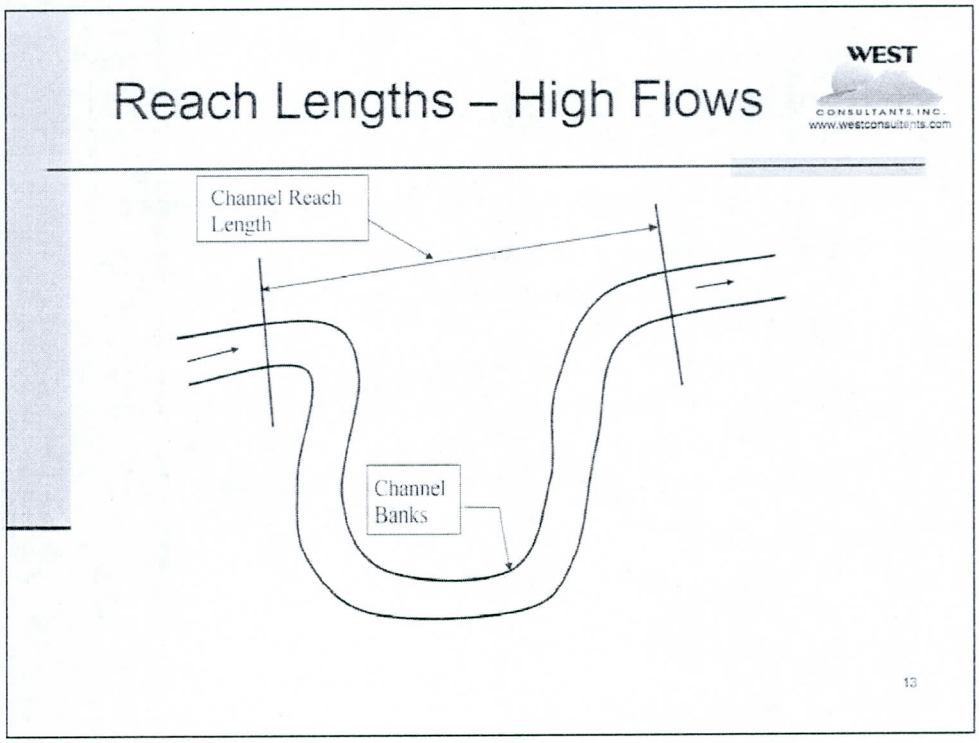


Reach lengths are estimated by the engineer. Typically, they are measured on a map using an engineer's scale along the estimated flow path between cross-sections. For the main channel, the reach length is measured along the thalweg. For the overbank areas, the reach lengths are measured along the centroid of the water in the overbank areas. The location of the center of mass of the overbank areas can be estimated based on the shape of the overbank areas. For example, if the overbank areas have an approximate rectangular cross-sectional area, then the centroid will be in the middle of the overbank area. If the overbank areas have an approximate triangular cross-sectional area, then the centroid will be about 1/3 of the distance from the bank station to the water's edge.

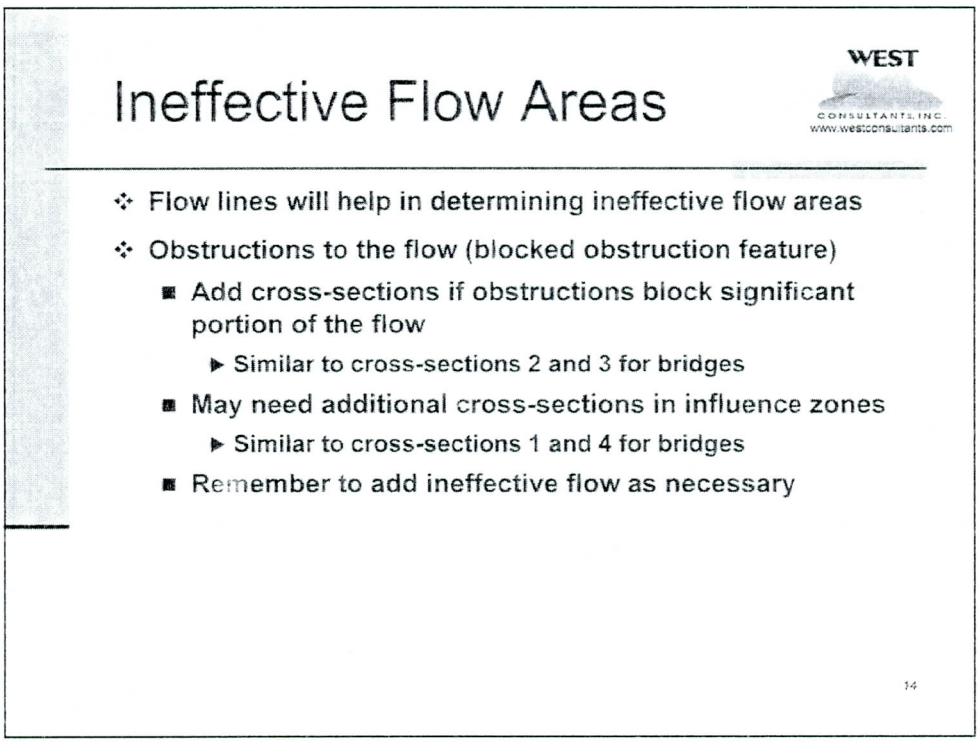
The reach lengths can be determined automatically using HEC GeoRAS. However, the engineer must still define a shape file that corresponds to the flow path for the main channel, the left overbank, and the right overbank.



For large flows on meandering streams, the overbanks may be overtopped and the channel flow may cut across the neck of the meander loop. In this case, the reach lengths (and cross-section locations) will be different than they are for low flows (see above figure). In this case, two geometry files are needed: one with the low flow cross-sections and reach lengths and one for the high flow cross-sections and reach lengths.



For large flows on meandering streams, the overbanks may be overtopped and the channel flow may cut across the neck of the meander loop. In this case, the reach lengths (and cross-section locations) will be different than they are for low flows (see above figure). In this case, two geometry files are needed: one with the low flow cross-sections and reach lengths and one for the high flow cross-sections and reach lengths.



Inspection of the flow lines will help in determining ineffective flow areas. In general, if the closest flow line to the boundary starts to significantly depart from the actual boundary, the area between the flow line and the boundary may be ineffective.

Identify those areas that contain obstructions to the flow. If these obstructions cover a significant portion of the projected flow area (length perpendicular to the flow lines), cross sections must be inserted at these locations.

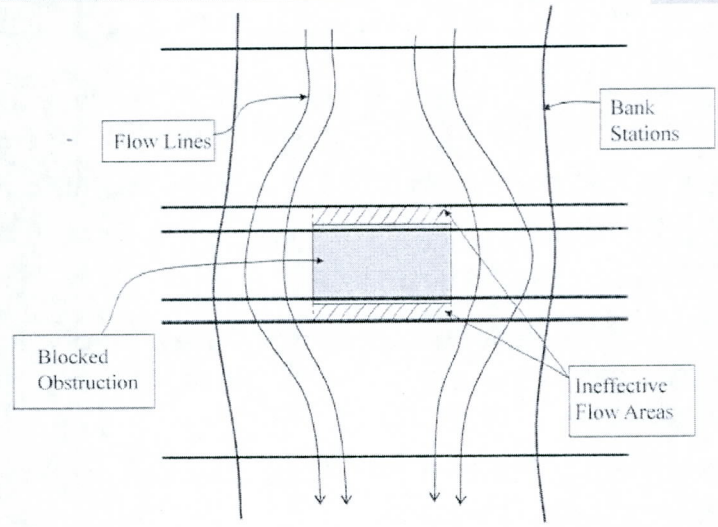
If you place additional cross sections in these areas to represent an obstruction, you should use the blocked obstruction option of HEC-RAS. Note that you should place cross sections in the influence zones upstream and downstream of these obstructions similar to cross sections 1 and 4 when modeling bridges. They should be located, again, using the 1 to 1 contraction ratio and 4 or 3 to 1 expansion ratio as rules of thumb.

Also, the blocked obstruction cross section should also have bounding cross sections similar to cross sections 2 and 3 in modeling bridges. Remember to add blocked ineffective flow areas at these cross sections.

You must also add cross sections upstream and downstream of the obstruction corresponding to cross sections 4 and 1 used in modeling bridges and culverts.

Ineffective Flow Areas

WEST CONSULTANTS, INC.
www.westconsultants.com



15

Cross-Section Spacing

WEST CONSULTANTS, INC. www.westconsultants.com

- Cross sections should be placed at representative locations to describe the changes in geometry .
- Samuels (1989) and Fread (2003) have equations for maximum spacing
- Other references (USGS, USACE)
- Additional cross sections should be added at locations where changes occur in discharge, slope, velocity, and roughness
- Cross sections must also be added at levees, bridges, culverts, and other structures

16

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

Other references that discuss cross-section spacing:

Davidian, J. (1984). "Computation of water-surface profiles in open channels." USGS Techniques of Water-Resources Investigations, Book 3, Chapter A15. http://water.usgs.gov/pubs/twri/twri3-a15/

Thompson, D.B., and Rogers, T.D. (1993). "Water surface profile computations – how many sections do I need?" Proc., ASCE Conference on Hydraulic Engineering, San Francisco, CA, July 25-30, 791-796.

USACE. (1986). "Accuracy of computed water surface profiles." Hydrologic Engineering Center, RD 26, Davis, CA.

Samuels' Equation for Cross-Section Spacing

WEST CONSULTANTS, INC.
www.westconsultants.com

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

Where: Δx is the cross section spacing distance;
D is the bankfull depth (m); and
S_0 is the bed slope

17

Cross-Section Spacing

WEST CONSULTANTS, INC.
www.westconsultants.com

- Major obstructions to flow (bridges, culverts, etc.)
- Stream gages/high water marks
- Embankments across the floodplain
- Significant changes in floodplain width
- Significant geometric changes in the channel
- Significant changes in roughness
- Significant changes in slope
- At or near levees
- Just upstream and downstream of tributaries
- At FEMA cross-section locations

18

Bed slope plays an important role in cross section spacing. Steeper slopes require more cross sections. Streams flowing at high velocities (i.e., streams with large slopes) may require cross sections on the order of 100 feet or less. Larger uniform rivers with flat slopes may only require cross sections on the order of 1000 ft or more.

Cross-Section Spacing

WEST
CONSULTANTS, INC.
www.westconsultants.com

- How do you know if you have enough?
  - Use the HEC-RAS cross section interpolation
  - Make a new plan and run the model
  - Compare the before and after
  - If no significant difference, then OK!

19

Cross-Section Spacing

WEST CONSULTANTS, INC.
www.westconsultants.com

- Check data collected by survey team
- Field notes
  - Check orientation
  - Check location
  - Compare elevations
  - Verify cross-sections were surveyed from left to right

20

Once the survey team has collected the topographic data needed to define the cross-sections, a check of the survey data should be performed. Using the field notes, check that the location and orientation of each cross-section is correct. Note contour elevations on the USGS quad sheets and compare them to the elevations recorded by the survey team. Also, verify that the cross-sections were survey from left to right looking downstream. If this is not the case, the stationing can be easily reversed in HEC-RAS.

Cross-Section Alignment

WEST CONSULTANTS, INC. www.westconsultants.com

- Skewed cross-sections (greater than 18°)
  - Resurvey cross-section
  - Reduced flow area using skew routine in RAS
- Meandering streams
  - Interpolation may require cross-section adjustments
  - Channel size may not change
  - Overbank areas may need to increase or decrease
- Dog-legged cross-sections
  - Allows cross-sections to always be perpendicular to flow lines

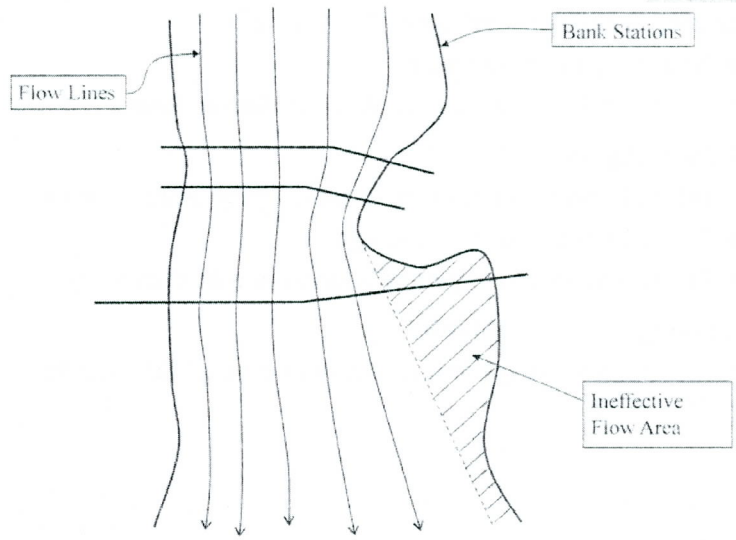
21

If the overall cross section is skewed more than 18 degrees from the perpendicular of the flow line (discussed later), either the cross section needs to be resurveyed or reduced by an appropriate multiplier.

Copying cross sections in a meandering stream may require adjustments (increasing or decreasing flow area) to the overbank areas but the channel size may not change.

Cross-Section Alignment

WEST CONSULTANTS, INC.
www.westconsultants.com



22

Discharge Data

WEST CONSULTANTS, INC. www.westconsultants.com

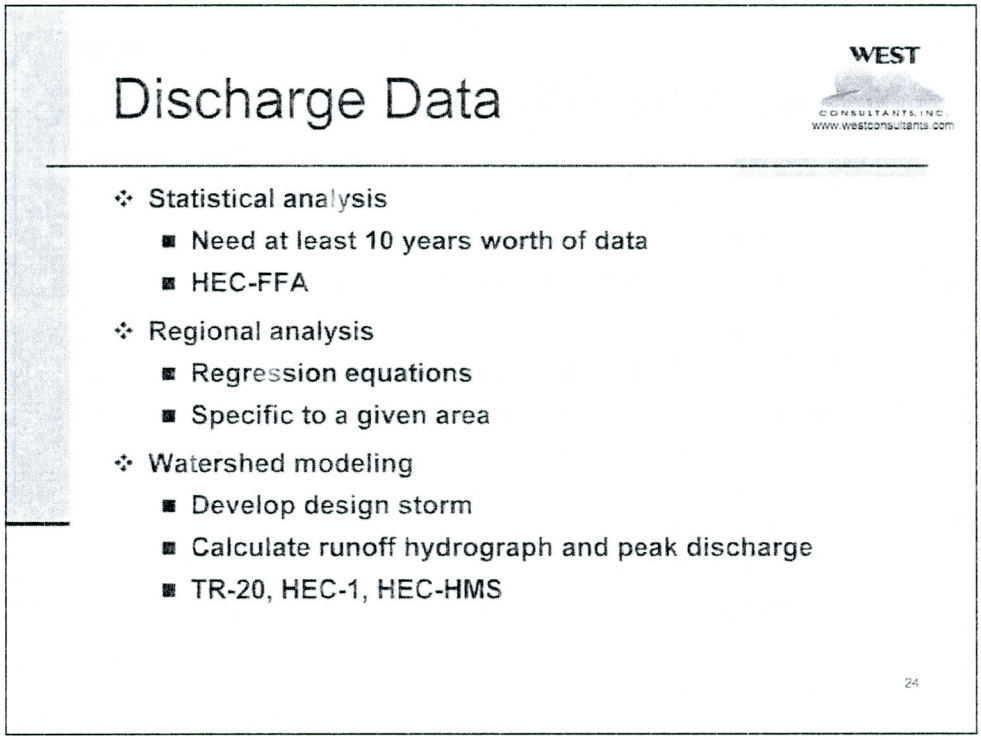
- Previous studies
  - Flood Insurance Studies (FIS)
  - Flood Control District's studies
- Gage Data
  - Single valued stage-discharge relation
  - Current meter discharge measurements
  - USGS rating system
    - Excellent rating is +/- 2%
    - Good rating is +/- 5%
    - Fair rating is +/- 8%
    - Poor rating is +/- 10%

23

Federal, state, and local agencies can be queried for previous reports and studies regarding your project area.

USGS classifies good flow measurements from Price current meters to be within ±5% of the true value. Some believe that this assumed error is optimistic. In any case, ±5%, on many river systems, translates into a stage error of ±1 foot. Acoustic velocity meters provide a continuous record (of velocity), but the current USGS technique calibrates these meters to reproduce measurements from Price current meters, so the AVM is as accurate as the current meter. Boat measurements are always suspect. Newer techniques using acoustic velocity meters with three beams mounted on boats are thought to be much better. Published discharge records should also be scrutinized. Continuous discharge is computed from discharge measurements, usually taken at bi-weekly or monthly intervals and the continuous stage record. The measurements are compiled into a rating curve and the departures of subsequent measurements from the rating curve are used to define shifts. The shifts are temporary changes in the rating curve due to unsteady flow effects (looped rating curve) and short term geomorphic changes. The quality of the record depends on the frequency of discharge measurements and the skill of the hydrologist. The only way to tell is to compare the discharge measurements to the flow record. Still, if the measurements are infrequent, one can only apply the flow record to the model and see how well the stage record is reproduced. Remember! Most published flow records are in mean daily flow. The modeler must somehow assign time values to these records (HEC).

(USGS, 1992) Office of Surface Water Technical Memorandum No 93.07, Policy Statement of Stage Accuracy, Dec. 4, 1992. http://water.usgs.gov/admin/memo/SW/sw93.07.html



Statistical analysis: statistical discharge-frequency estimates are generally considered reasonably accurate only for frequencies that are similar to the length of the actual record. A "rule of thumb" states that frequency estimates should extend to less than twice the period of record. In addition, the data must be homogenous. The HEC-FFA program can be used as a tool to perform a statistical analysis.

Regional analysis: If the study requires only peak discharges and floodplain or reservoir storage is not a factor, then regional analysis is often used. Regression equations are often considered the next most accurate estimate of peak-discharge frequency, because the equations are derived from statistical analysis of actual gage data. Regional analysis has been used by USGS to develop flood-peak prediction equations for states, urban areas around the country as well as for specific cities.

Watershed Modeling: Only watershed modeling gives full discharge hydrographs and translates these hydrographs through the watershed, allowing the effects of timing, tributary inflows, reservoirs, channel modifications, and other flood mitigation components to be properly evaluated. Hydrologic modeling develops an actual or synthetic design storm and calculate the runoff hydrograph and peak discharges for the specified event. These models includes TR-20, HEC-1, and HEC-HMS.

Roughness Data

WEST CONSULTANTS, INC. www.westconsultants.com

- Judgment and/or experience
- Table lookup
- Picture comparison
- Cowan's equation

$$n = (n_0 + n_1 + n_2 + n_3 + n_4)m_5$$

25

Judgment/Experience: Engineers with open channel hydraulic studies experiences can develop an intuitive feel for the appropriate Manning's n values. However, one should always evaluate n values with many different techniques before adopting a value, instead of relying on his/her own experience or judgment.

Table Lookup: Through site visits, the study channel or floodplain can be described and then compared to standard descriptions of channel or floodplain conditions already defined in different hydraulic tables (e.g., Chow, 1959).

Picture Comparison: If gage information for the site is not available, comparing pictures of the study reach to similar reaches whose channel n values has been determined can be a very accurate method for estimating n values.

Cowan's Equation: The Cowan Equation (1956) is a very useful method to deriving an analytic estimate of channel n values for the study reach. Since Cowan developed his procedure from studying 40 to 50 small- to moderate-size channels, this equation is questionable for streams with a hydraulic radius larger than 15 ft (4.6m).

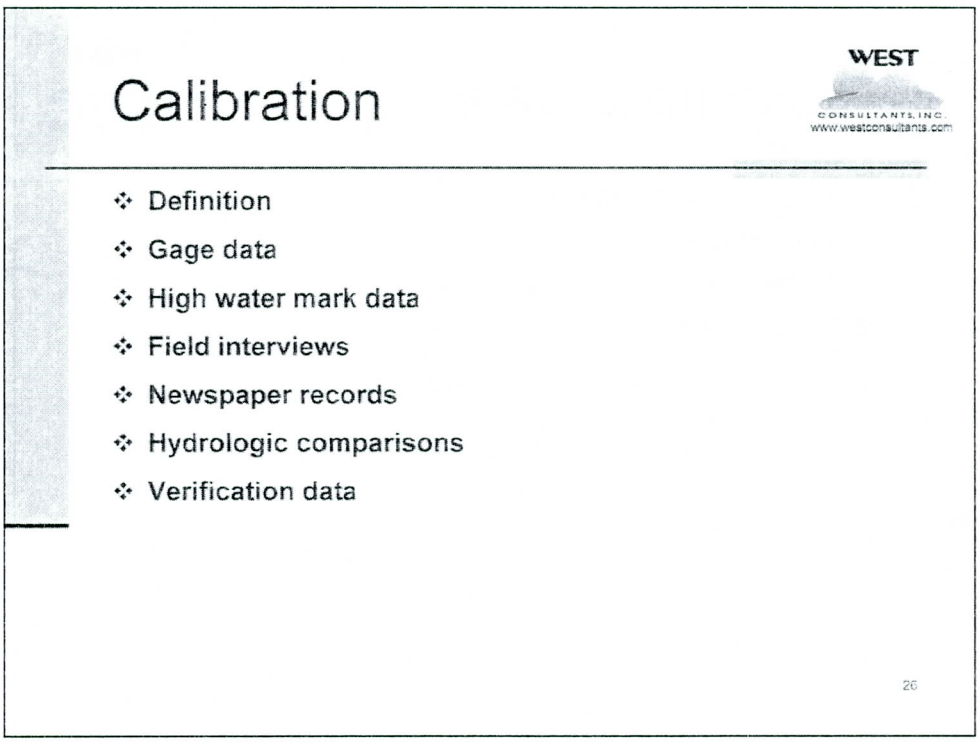
n = overall roughness value

n0 = base n-value for a smooth, straight reach

n1 = additional roughness due to surface irregularities

n2 = additional roughness due to variations of the shape and size of the channel cross-sections

n3 = additional roughness due to obstructions



Calibration Definition: The hydraulic model always requires calibration to give a sense of confidence that the program output does properly represent what would occur on the prototype river. Calibration adjusts model parameters (within reason) so that predicted system performance agrees with observed system performance over a range of conditions.

Gage Data: Calibrating a model to only a single cross section has limited value. Output from the floodplain modeling program should be calibrated to several actual discharge and stage data pairs representing different runoff events. If multiple gage sites or flood high water marks are available, they should be used to calibrate the model to several sites along the stream.

High water Mark Data: If one or more large floods have occurred, high water marks records are often available to calibrate the model. The high water marks can contain some error. High water marks obtained from field interviews depend on a person's memory and whether the individual witnessed the flood level on the recession side rather than at the peak.

Field Interviews: If no formal high water mark surveys were not available, the engineer can perform a survey to local residents to obtain these estimates directly.

Newspaper Records: Because references to flood level on a local landmark are often printed, so newspaper records are often a good source of high water mark data.

Consistency Checks

WEST CONSULTANTS, INC. www.westconsultants.com

- Velocities and flow percentage
  - Main channel
  - Overbank areas.
- Should be fairly consistent
- If not consistent
  - Is there a physical reason for the changes?
  - If not, the model must be adjusted

27

Look at the velocities and flow percentage of the main channel and overbank areas. If the prototype is fairly uniform in the longitudinal direction, the velocities and flow distribution of the channel and overbanks should be fairly consistent also.

If the model shows severe changes occurring from cross section to cross section, there should be a physical reason for the changes; if not, the model must be adjusted.

Consistency Checks

WEST CONSULTANTS, INC. www.westconsultants.com

- Example 1 – Consistency in velocity
  - Cross section 1: 2 m/s
  - Cross section 2: 0.5 m/s
  - Cross section 3: 3 m/s
  - Channel geometry and slope is relatively uniform
- Not a reasonable result!
- Need to find out why there is an inconsistency

28

For instance, if channel velocity goes alternately 2 m/s to 0.5 m/s to 3 m/s in the downstream direction and the channel geometry and slope is relatively the same throughout the reach, this is not a reasonable model result.

It may be that where the channel velocity was 0.5 m/s, there was a large increase in overbank flow area or conveyance, thus decreasing the channel discharge, resulting is a lower channel velocity. One may then consider if all of the overbank area should be available for conveyance in such a short distance.

Consistency Checks

WEST CONSULTANTS, INC. www.westconsultants.com

- Example 2 – Consistency in flow
  - Percentage of discharge in one overbank
  - See figure on next slide
- Is reach length between the cross sections long enough?
- Fixes
  - Use ineffective flow areas
  - Adjust the overbank roughness values
- Remember to check the channel discharge and velocity in adjacent cross sections

29

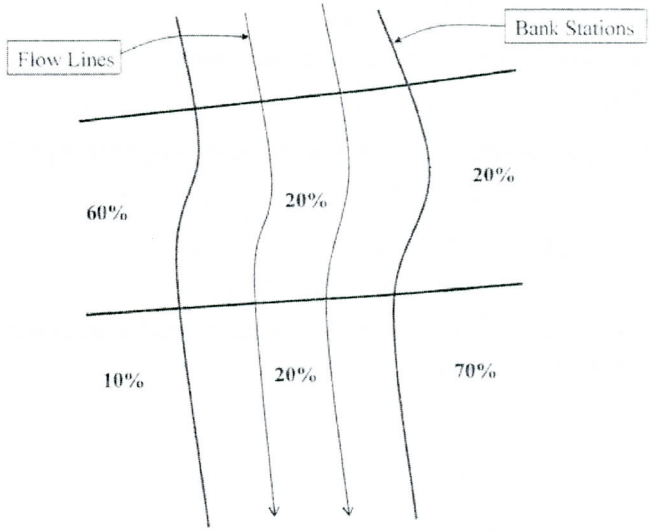
Another example would be if the percentage of discharge (say 60%) in one overbank changed to 10% in the next downstream cross section and the percentage in the channel stayed the same (say 20%). This means that the other overbank changed from 20% to 70% in the downstream direction.

You must judge if the reach length between the cross sections was sufficiently long to allow such a large transfer of water from one overbank, through the channel, and to the other overbank.

Appropriate adjustments would be to use the ineffective flow options or adjust the overbank "n" values. Again, remember to check the channel discharge and velocity in relation to the adjacent cross sections.

Consistency Checks

WEST CONSULTANTS, INC.
www.westconsultants.com



30

WEST

CONSULTANTS, INC.
www.westconsultants.com

HEC-RAS, River Analysis System

By WEST Consultants, Inc.

Optional Capabilities of HEC-RAS

Note: This lecture was partially derived from Chapter 4 of the HEC-RAS Hydraulic Reference Manual.

This document is property of WEST Consultants, Inc., and is not to be duplicated without prior consent

Overview

WEST CONSULTANTS, INC.
www.westconsultants.com

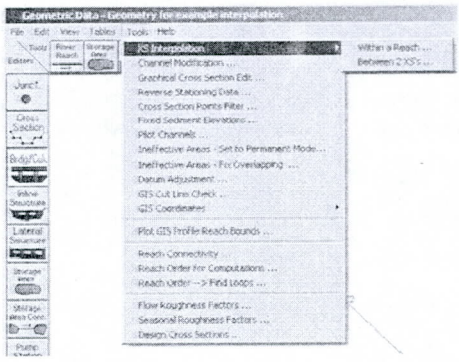
- Cross-section interpolation
- Mixed flow
- Momentum equation for junctions
- Flow optimization
- Flow distribution
- Water quality analysis

2

Cross Section Interpolation

WEST CONSULTANTS, INC.
www.westconsultants.com

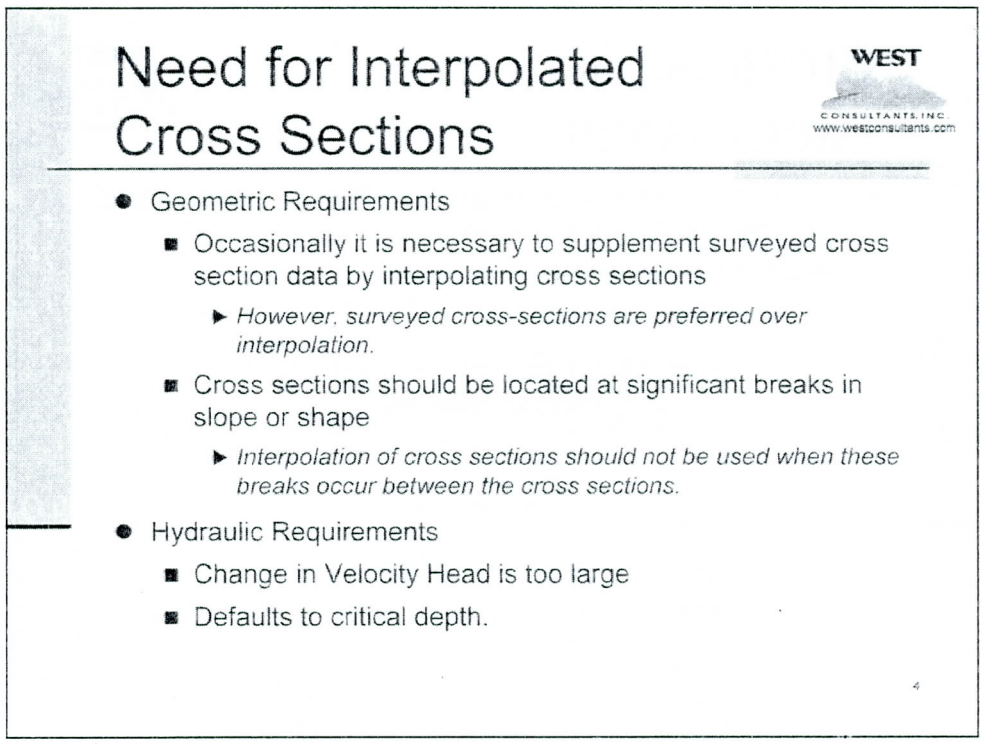
- Cross Section interpolation is activated from the Tools menu of the Geometric Data editor screen.
  - Need for interpolated Cross Sections
  - String Model Concepts



Cross Section interpolation is activated from the **Tools** menu of the Geometric Data editor screen.

Need for interpolated Cross Sections

String Model Concepts



Geometric Requirements

Occasionally it is necessary to supplement surveyed cross section data by interpolating cross sections in between two surveyed sections.

Cross sections should be located at significant breaks in slope or shape; therefore, interpolation of cross sections should not be used when these breaks occur between the cross sections.

Hydraulic Requirements

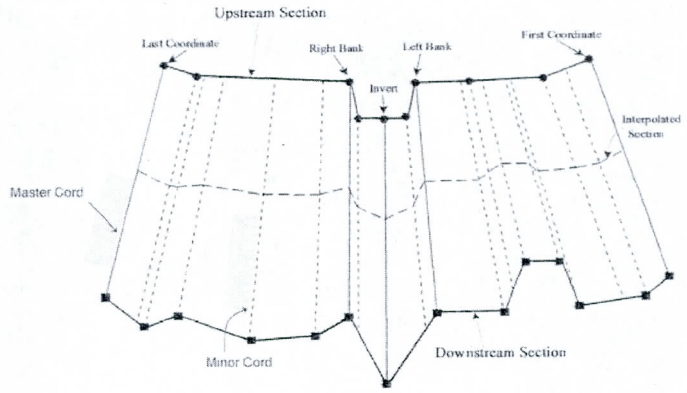
Interpolated cross sections are often required when the change in velocity head is too large to accurately determine the change in the energy gradient.

When cross sections are spaced too far apart, the program may end up defaulting to critical depth.

String Model Concepts

WEST CONSULTANTS, INC.
www.westconsultants.com

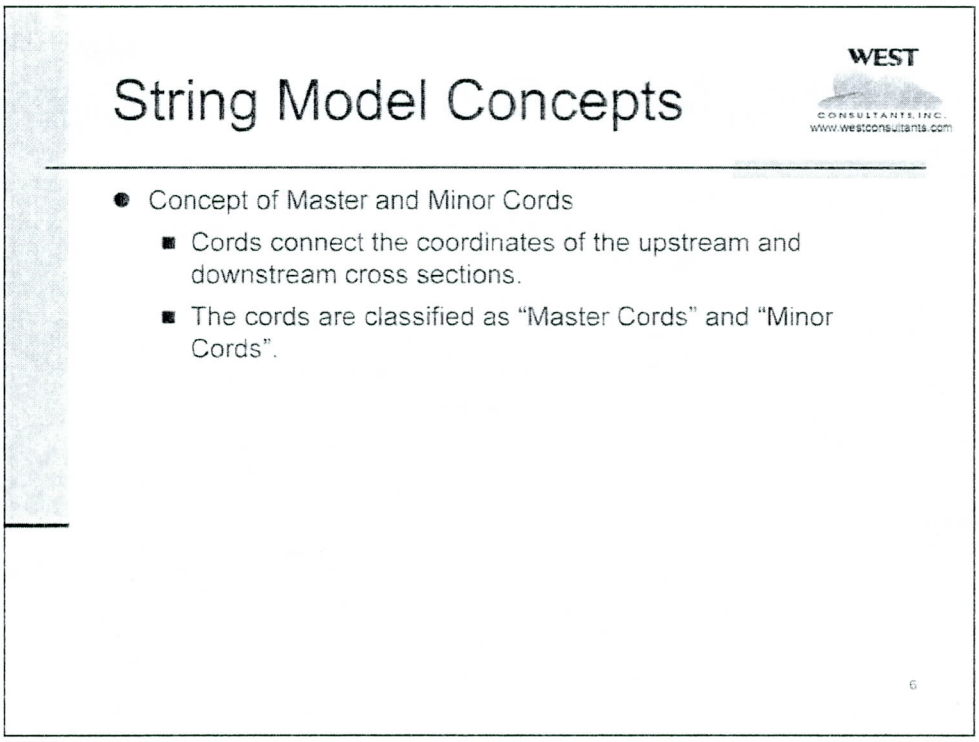
- The HEC-RAS program has the ability to generate cross sections by interpolating the geometry between two user-entered cross sections.



5

General

The HEC-RAS program has the ability to generate cross sections by interpolating the geometry between two user-entered cross sections.



Concept of Master and Minor Cords

The string model in HEC-RAS consists of cords that connect the coordinates of the upstream and downstream cross sections.

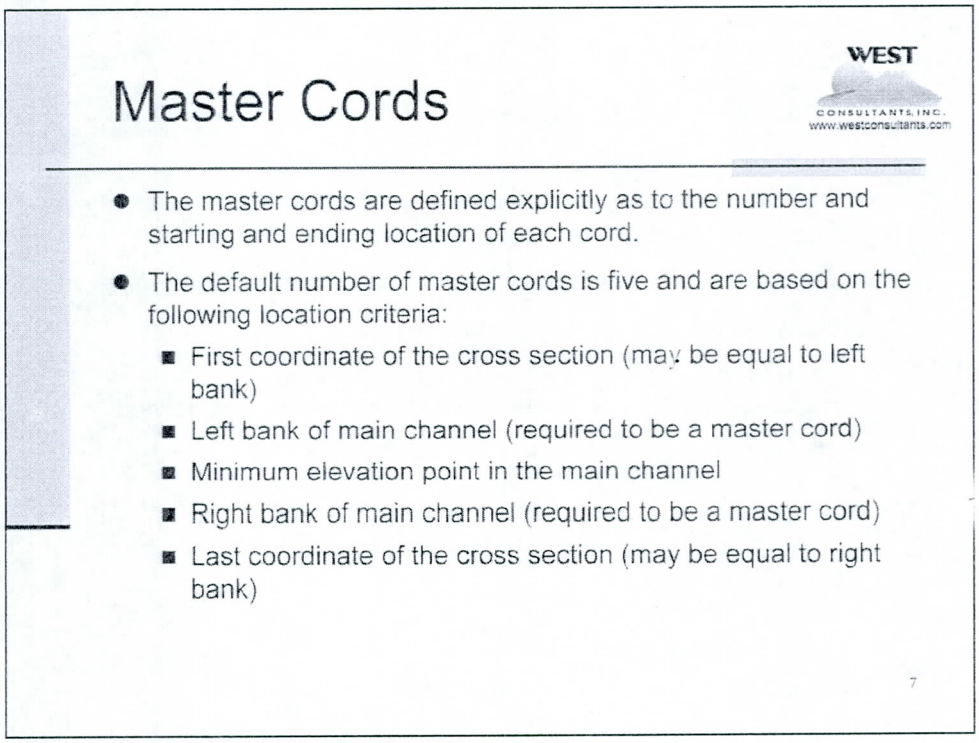
The cords are classified as "Master Cords" and "Minor Cords".

Master Cords

The master cords are defined explicitly as to the number and starting and ending location of each cord.

The default number of master cords is five and are based on the following location criteria:

First coordinate of the cross section (may be equal to left bank)
Left bank of main channel (required to be a master cord)
Minimum elevation point in the main channel
Right bank of main channel (required to be a master cord)
Last coordinate of the cross section (may be equal to right bank)



Concept of Master and Minor Cords

The string model in HEC-RAS consists of cords that connect the coordinates of the upstream and downstream cross sections.

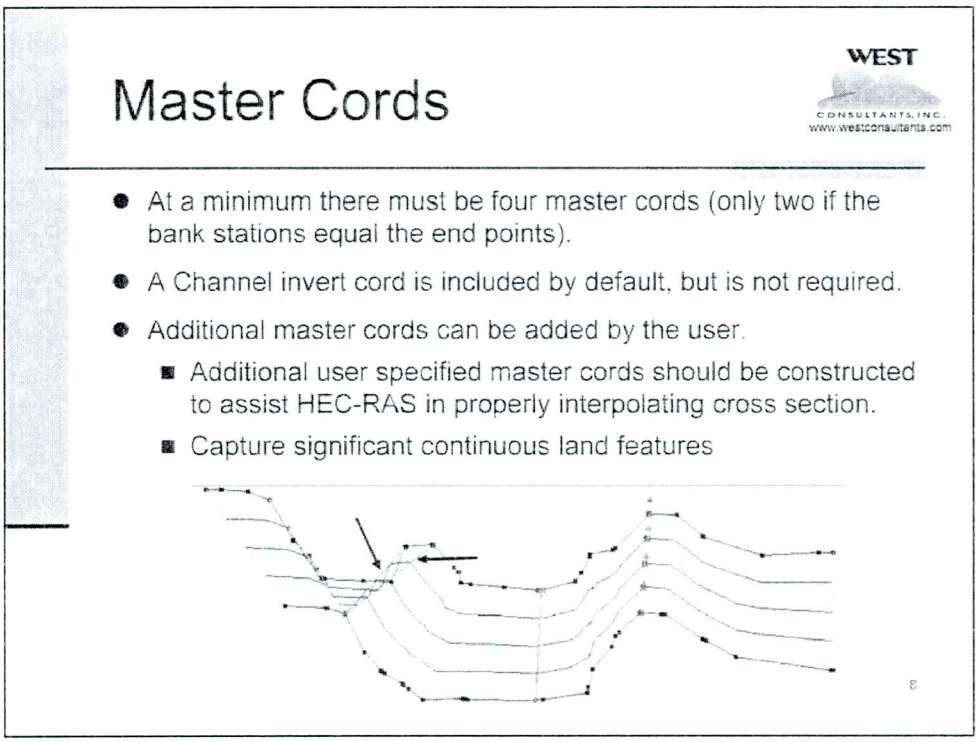
The cords are classified as "Master Cords" and "Minor Cords".

Master Cords

The master cords are defined explicitly as to the number and starting and ending location of each cord.

The default number of master cords is five and are based on the following location criteria:

First coordinate of the cross section (may be equal to left bank)
Left bank of main channel (required to be a master cord)
Minimum elevation point in the main channel
Right bank of main channel (required to be a master cord)
Last coordinate of the cross section (may be equal to right bank)



At a minimum there must be two master cords, but there is no maximum; therefore, additional master cords can be added by the user.

Additional user specified master cords should be constructed to assist HEC-RAS in properly interpolating cross section.

This is particularly important if there are important continuous land features from one cross section to another that are not the same distance from the cross section edge or channel from one cross section to another.

This is explained in Chapter 6 of the HEC-RAS User's Manual, under cross section Interpolation.

Minor Cords

WEST CONSULTANTS, INC. www.westconsultants.com

- Minor cords are automatically generated.
- Determined by establishing coordinates between the upstream and downstream cross-sections
- The station values at the opposite cross section are determined by proportioning.
- Station and Elevation at the new XS are then Interpolated.
- The number of minor cords will be equal to the sum of all the coordinates in the upstream and downstream sections minus the number of master cords.

9

Minor Cords

The minor cords, automatically generated by the interpolation routines, are determined by taking an existing coordinate in either the upstream or downstream section and establishing a corresponding coordinate at the opposite cross section by either matching an existing coordinate or interpolating one.

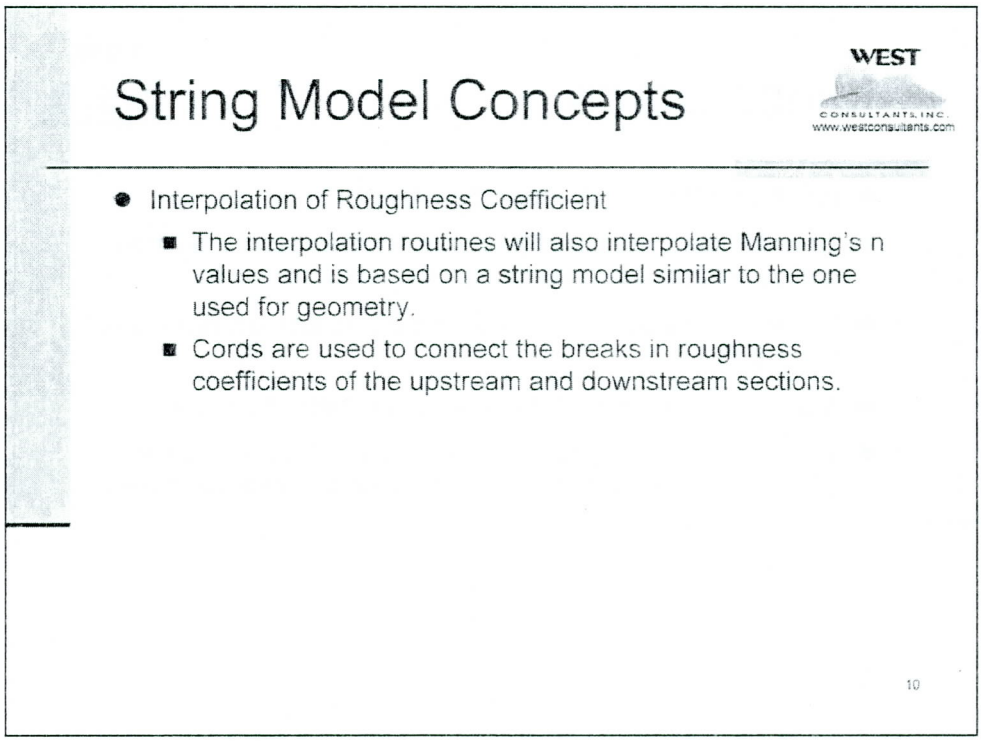
The station values at the opposite cross section is determined by computing the proportional distance that the known coordinate represents between master cords and then applying the proportion to the distance between master cords of the opposite section.

The number of minor cords will be equal to the sum of all the coordinates in the upstream and downstream sections minus the number of master cords.

Once all the minor cords are computed, the routines can then interpolate any number of sections between the two known cross sections.

Interpolation is accomplished by linearly interpolating between the elevations at the ends of a cord.

Interpolated points are generated at all of the minor and master cords.



Interpolation of Roughness Coefficient

The interpolation routines will also interpolate roughness coefficients (Manning's n values) and is based on a string model similar to the one used for geometry.

Cords are used to connect the breaks in roughness coefficients of the upstream and downstream sections.

Interpolation of Roughness Coefficient

WEST CONSULTANTS, INC. www.westconsultants.com

- The default number of master cords is four and are based on the following location criteria:
  - First coordinate of the cross section (may be equal to left bank)
  - Left bank of main channel
  - Right bank of main channel
  - Last coordinate of the cross section
- When either of the two cross sections has more than three n values, additional minor cords are added at all other n value break points.
- Interpolation of roughness coefficients is then accomplished in the same manner as geometry interpolation.

11

The default number of master cords is four and are based on the following location criteria:

First coordinate of the cross section (may be equal to left bank)

Left bank of main channel

Right bank of main channel

Last coordinate of the cross section

When either of the two cross sections has more than three n values, additional minor cords are added at all other n value break points.

Interpolation of roughness coefficients is then accomplished in the same manner as geometry interpolation.

Mixed Flow Regime Calculations

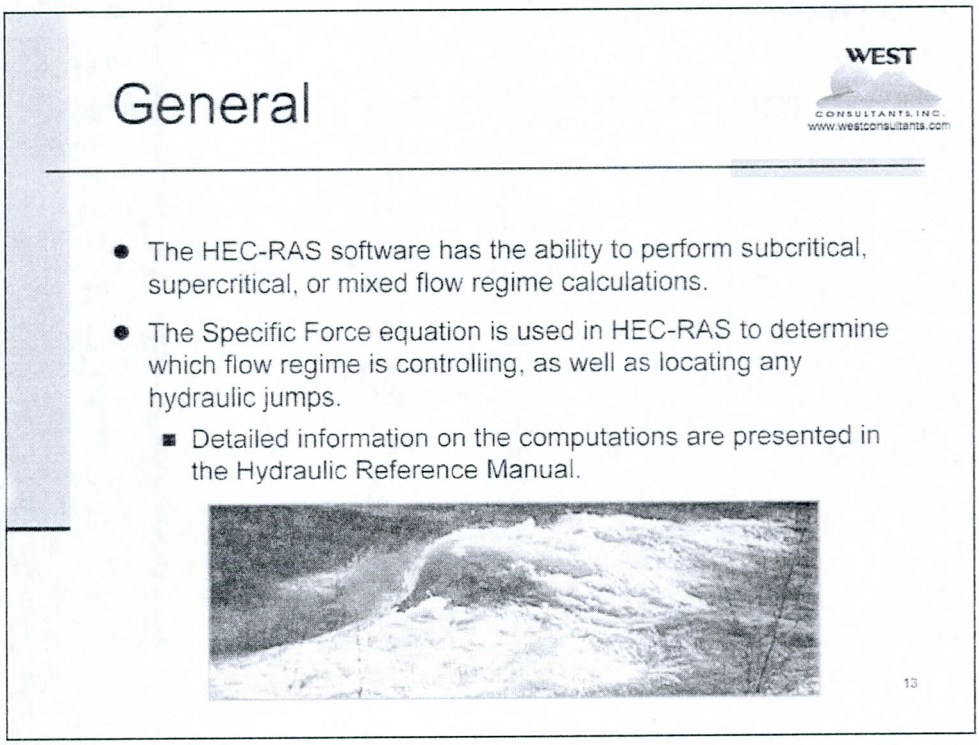
WEST CONSULTANTS, INC.
www.westconsultants.com

- General
- Calculation Scheme

12

General

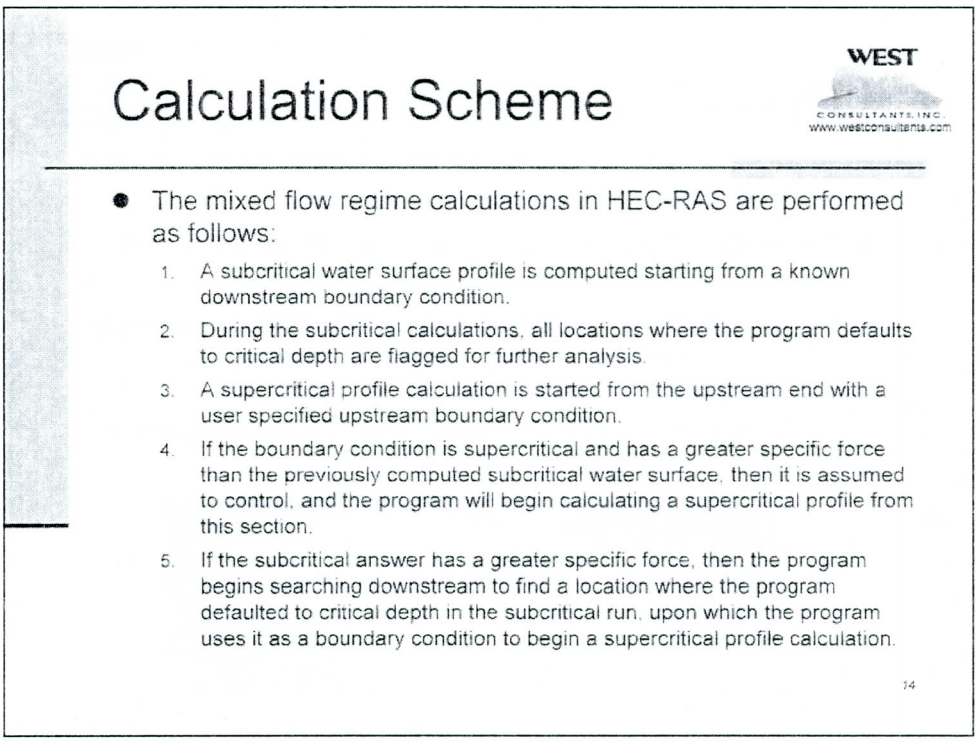
Calculation Scheme



The HEC-RAS software has the ability to perform subcritical, supercritical, or mixed flow regime calculations.

The Specific Force equation is used in HEC-RAS to determine which flow regime is controlling, as well as locating any hydraulic jumps.

Detailed information on the computations are presented in Chapter 4, pages 4-6 to 4-8 of the Hydraulic Reference Manual



The mixed flow regime calculations in HEC-RAS are performed as follows:

A subcritical water surface profile is computed starting from a known downstream boundary condition.

During the subcritical calculations, all locations where the program defaults to critical depth are flagged for further analysis.

A supercritical profile calculation is started from the upstream end with a user specified upstream boundary condition.

If the boundary condition is supercritical and has a greater specific force than the previously computed subcritical water surface, then it is assumed to control, and the program will begin calculating a supercritical profile from this section.

If the subcritical answer has a greater specific force, then the program begins searching downstream to find a location where the program defaulted to critical depth in the subcritical run, upon which the program uses it as a boundary condition to begin a supercritical profile calculation.

Calculation Scheme

WEST CONSULTANTS, INC. www.westconsultants.com

6. A supercritical profile in the downstream direction is calculated until it reaches a cross section that has valid subcritical and supercritical answers, calculates the specific force of both computed water surface elevations and selects the one with the greater specific force.
7. If the supercritical answer has a greater specific force, the program continues making supercritical calculations in the downstream direction and comparing the specific forces
8. When the program reaches a cross section whose subcritical answer has a greater specific force than the supercritical answer, the program assumes that a hydraulic jump occurred between that section and the previous cross section.
9. The program then goes to the next downstream location that has a critical depth answer and continues the process.

15

A supercritical profile in the downstream direction is calculated until it reaches a cross section that has valid subcritical and supercritical answers, calculates the specific force of both computed water surface elevations and selects the one with the greater specific force.

If the supercritical answer has a greater specific force, the program continues making supercritical calculations in the downstream direction and comparing the specific forces

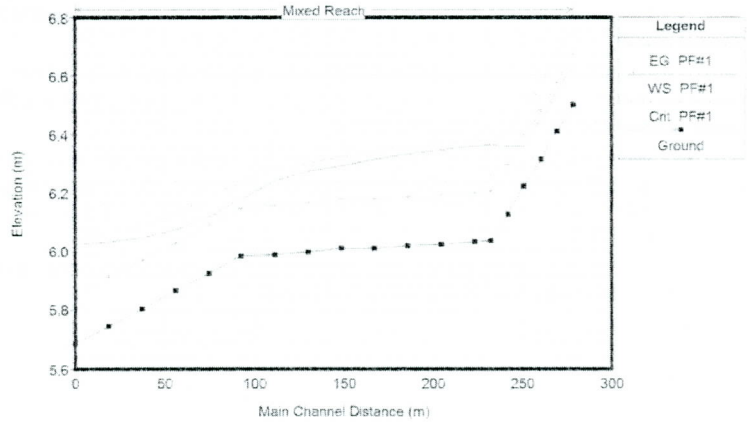
When the program reaches a cross section whose subcritical answer has a greater specific force than the supercritical answer, the program assumes that a hydraulic jump occurred between that section and the previous cross section.

The program then goes to the next downstream location that has a critical depth answer and continues the process.

Calculation Scheme

WEST CONSULTANTS, INC.
www.westconsultants.com

Examples of Mixed Flow



16

Modeling Stream Junctions

WEST CONSULTANTS, INC.
www.westconsultants.com

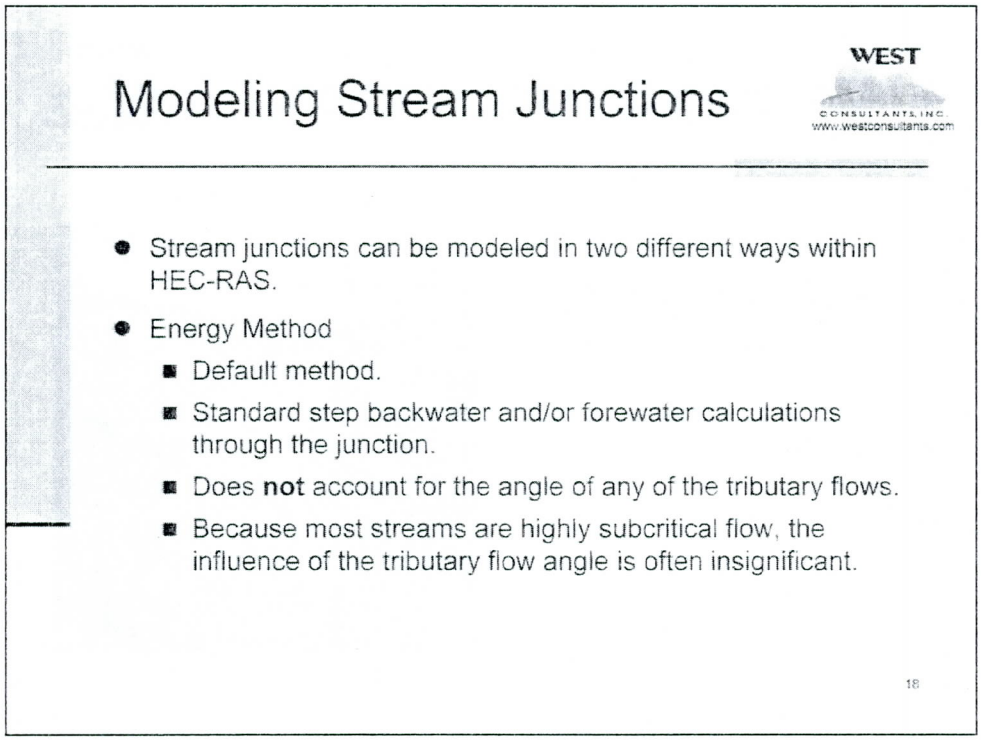
- General
- Possible Junction Conditions
- Momentum Method

17

General

Possible Junction Conditions

Momentum Method



Stream junctions can be modeled in two different ways within HEC-RAS.

Energy Method

The default method is an energy based solution.

The energy method solves for water surfaces across the junction by performing standard step backwater and forewater calculations through the junction.

The energy method does not account for the angle of any of the tributary flows.

Because most streams are highly subcritical flow, the influence of the tributary flow angle is often insignificant.

Modeling Stream Junctions

WEST CONSULTANTS, INC. www.westconsultants.com

- Momentum Method
  - If the angle of the tributary plays an important role in influencing the water surface around the junction.
  - 1-D form of the momentum equation, angles of the tribs are used to evaluate the forces associated with the trib flows.

19

Momentum Method

If the angle of the tributary plays an important role in influencing the water surface around the junction, then the user should switch to the alternative method available in HEC-RAS, which is a momentum based method.

The momentum based method is a one dimensional formulation of the momentum equation, but the angles of the tributaries are used to evaluate the forces associated with the tributary flows.

Possible Junction Conditions

WEST CONSULTANTS, INC. www.westconsultants.com

- There are six possible flow conditions that HEC-RAS can handle at a junction
  - Subcritical flow – flow combining
  - Subcritical flow – flow split
  - Supercritical flow – flow combining
  - Supercritical flow – flow split
  - Mixed flow regime – flow combining
  - Mixed flow regime – flow split
- The most common situations are the subcritical flow cases (1) and (2).

20

There are six possible flow conditions that HEC-RAS can handle at a junction

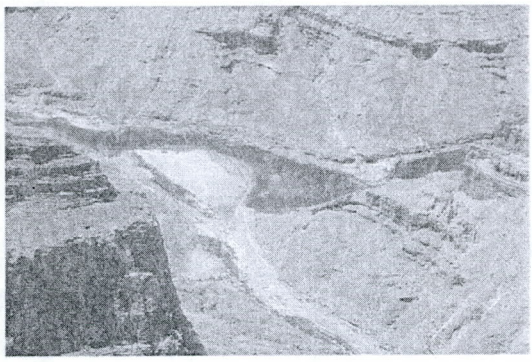
Subcritical flow – flow combining
Subcritical flow – flow split
Supercritical flow – flow combining
Supercritical flow – flow split
Mixed flow regime – flow combining
Mixed flow regime – flow split

The most common situations are the subcritical flow cases (1) and (2).

Momentum Calculation at Junctions

WEST CONSULTANTS, INC. www.westconsultants.com

- Required Data
  - User must supply the angle for any reach with flow lines that are not parallel to the mainstem's flow lines.



Careful! Angles may be different between low and high flow events.

21

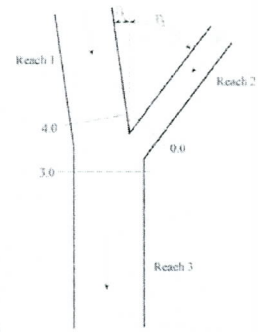
Required Data

To use the momentum based method, the user must supply the angle for any reach with flow lines that are not parallel to the main stem's flow lines.

Momentum Calculation at Junctions

WEST CONSULTANTS, INC. www.westconsultants.com

- "Combining" Junctions
  - Angles for both reaches 1 (θ1) and 2 (θ2) should be entered.
  - Each angle is taken from a line that is perpendicular to cross section 3.0 of reach 3 (the mainstem.)
- For subcritical flow, the water surface is computed up to section 3.0 of reach 3 by normal standard step backwater calculations.
- If the momentum equation is selected, the program solves for the water surfaces at sections 4.0 and 0.0 by performing a momentum balance across the junction.



22

Combining Junctions

An example of a flow combining junction is shown in Figure 3.

In this example, angles for both reaches 1 (Θ_1) and 2 (Θ_2) should be entered.

Each angle is taken from a line that is perpendicular to cross section 3.0 of reach 3 (the main stem.)

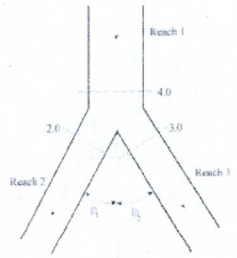
For subcritical flow, the water surface is computed up to section 3.0 of reach 3 by normal standard step backwater calculations.

If the momentum equation is selected, the program solves for the water surfaces at sections 4.0 and 0.0 by performing a momentum balance across the junction

Momentum Calculation at Junctions

WEST CONSULTANTS, INC. www.westconsultants.com

- Subcritical Flow at "Split" Junctions
  - For subcritical flow, the water surface elevation is known at sections 2.0 and 3.0, and the water surface elevation at section 4.0 can be found.



23

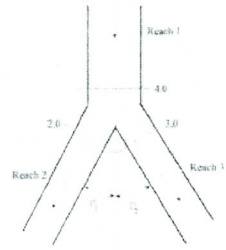
Subcritical Flow at Split Junctions

For subcritical flow, the water surface elevation is known at sections 2.0 and 3.0, and the water surface elevation at section 4.0 can be found.

Momentum Calculation at Junctions

WEST CONSULTANTS, INC.
www.westconsultants.com

- Supercritical Flow at "Split" Junctions
  - For supercritical flow, the water surface is known at section 4.0 only, and, therefore, the water surface elevation at section 3.0 and 2.0 must be solved simultaneously.
- Mixed Flow
  - Mixed flow regime computations for a flow split are handled in the same manner as the energy based solution, except the momentum equation is used to solve for the water surface elevations across the junction.



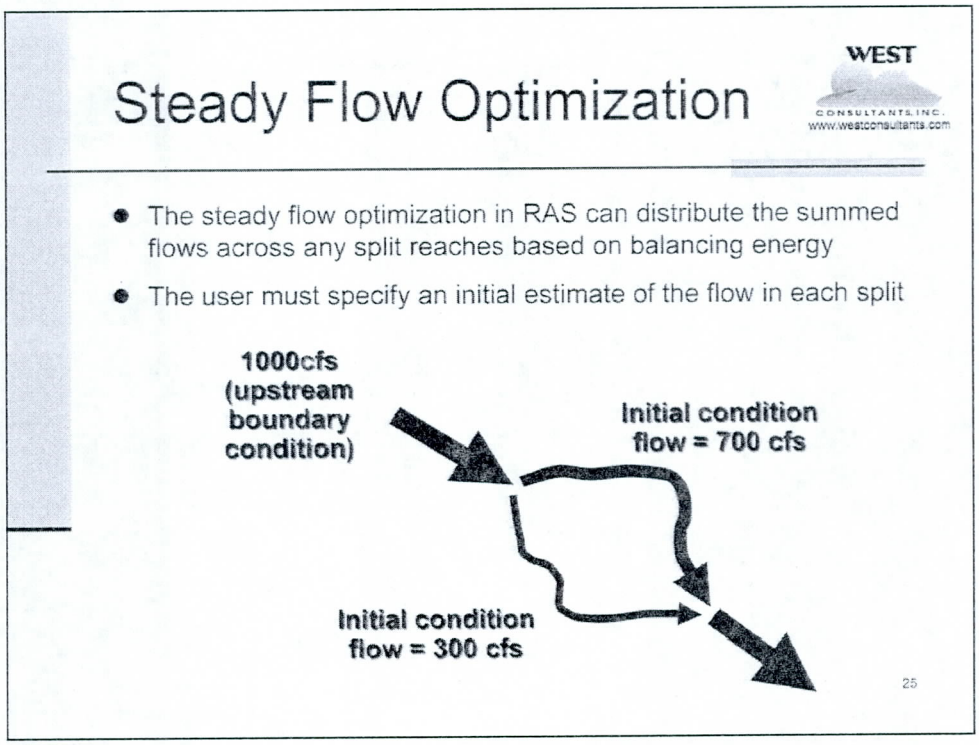
24

Supercritical Flow at Split Junctions

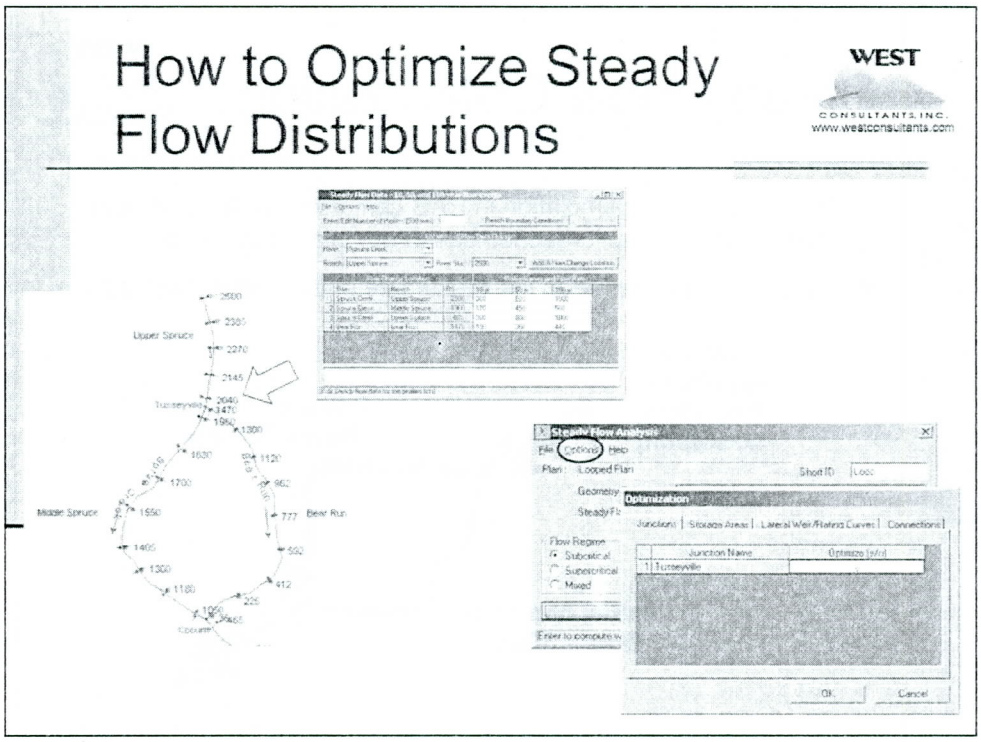
For supercritical flow, the water surface is known at section 4.0 only, and, therefore, the water surface elevation at section 3.0 and 2.0 must be solved simultaneously.

Mixed Flow

Mixed flow regime computations for a flow split are handled in the same manner as the energy based solution, except the momentum equation is used to solve for the water surface elevations across the junction.



The steady flow optimization in RAS will distribute the summed flows across any split reaches based on balancing energy. For the example in the slide, RAS will iterate until it obtains a solution to where the energy at the upstream cross sections of the two split channels are equal, and the summed flow between the two channels is 1000 cfs (300 cfs + 700 cfs). The sum of the flows across the split reaches does not need to equal the flows in the combined upstream or downstream reaches. For example, the upstream reach in the slide could have 800 cfs, and the downstream reach 1200 cfs. In this case, RAS will still distribute a total of 1000 cfs between the two split channels.

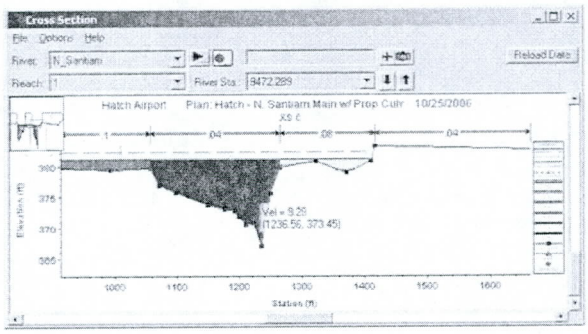


Select **Flow Optimizations** from the **Options** window in the Steady Flow Analysis Editor. Type a "y" at the junction(s), or other location, where you wish to optimize flow (HEC). Remember, in a subcritical backwater analysis, energy is balanced across the two downstream channels at a flow split, and not at a downstream confluence. In the example in the slide, assuming a subcritical run, optimization would then need to occur at the Tusseyville junction and not at the Coburn junction. The Beta Version of RAS allows optimization for supercritical flow.

Flow Distribution Calculations

WEST CONSULTANTS, INC.
www.westconsultants.com

- Default Flow Distribution
- Defining Distribution Locations
- Computations
- Cautions in Using the Flow Distribution Option



27

Default Flow Distribution

Defining Distribution Locations

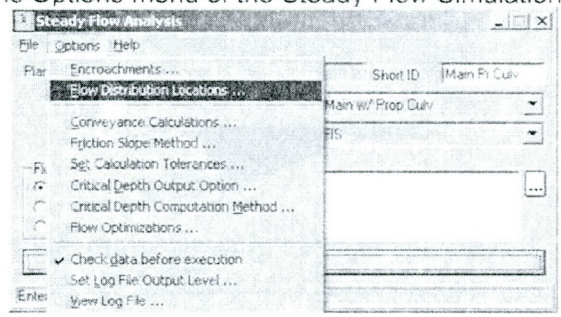
Computations

Cautions in Using the Flow Distribution Option

Default Flow Distribution

WEST CONSULTANTS, INC. www.westconsultants.com

- The general cross section output shows the distributions of flow in three subdivisions of the cross section.
- Additional output showing the distribution of flow for multiple subdivisions
  - Through the Options menu of the Steady Flow Simulation window.



28

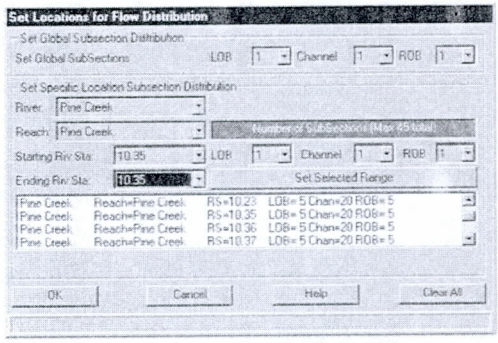
The general cross section output shows the distributions of flow in three subdivisions of the cross section.

Additional output showing the distribution of flow for multiple subdivisions of the left and right overbanks, as well as the main channel, can be requested by the user through **Options** menu of the Steady Flow Simulation window.

Defining Distribution Locations

WEST CONSULTANTS, INC.
www.westconsultants.com

- Specific locations or all locations in the model.
- Define Slices for left overbank, main channel, and right overbank
  - Up to 45 total slices
- Each sub-section must have at least one slice.



29

The user can select specific locations or all locations in the model.

The number of slices for the flow distribution computations must be defined for the left overbank, main channel, and the right overbank with up to 45 total slices which can vary from cross section to cross section.

Each flow element (left overbank, main channel, and right overbank) must have at least one slice.

Flow Distribution Computations

WEST CONSULTANTS, INC. www.westconsultants.com

- The computations for the flow distribution are performed after Steady Flow computations
  - Water surface elevation is computed using the three flow subdivisions and balancing the energy equation.
  - The program computes an area, wetted perimeter, and hydraulic depth (area over top width) for each user defined slice.
  - Using the originally computed energy slope (Sf), the conveyance and percentage of discharge are computed for each of the slices.

30

For further details on how to request and view flow distribution output, see Chapters 7, 8, and 9 of the HEC-RAS User's Manual.

The computations for the flow distribution are performed after the program has calculated a water surface elevation and energy. The flow distribution computations are performed as follows:

> Water surface elevation is computed using the three flow subdivisions and balancing the energy equation.
>
> The program computes an area, wetted perimeter, and hydraulic depth (area over top width) for each user defined slice.
>
> Using the originally computed energy slope (S_f), the conveyance and percentage of discharge are computed for each of the slices.

Flow Distribution Computations

WEST CONSULTANTS, INC. www.westconsultants.com

- **The computed conveyance for each slice are summed**. The sum of the slice-computed conveyances is usually greater than the original conveyance
- A **ratio of the original total conveyance** (from the normal calculations) to the total slice conveyance **is determined**.
- The **ratio is applied to each of the slices**, in order to achieve the same conveyance as was originally computed.
- An **average velocity** for each slice is computed by taking the **same conveyance as was originally computed**.

31

The computed conveyance for each slice are summed. The sum of the slice computed conveyance is usually greater than the original conveyance because the computed conveyance for a given water surface elevation will increase with each new slice.

A ratio of the original total conveyance (from the normal calculations) divided by the total slice conveyance is determined.

The ratio is applied to each of the slices, in order to achieve the same conveyance as was originally computed.

An average velocity for each slice is computed by taking the same conveyance as was originally computed.

Flow Distribution Cross Section Output Table

WEST CONSULTANTS, INC.
www.westconsultants.com

Flow Distribution Output

File Type Options Help

River: Pine Creek Profile: PF 1

Reach: Pine Creek Riv Sta: 10.90

Left Sta (ft)	Right Sta (ft)	Flow (cfs)	Area (sq ft)	W/P (ft)	% Conv	Hydr D. (ft)	Velocity (ft/s)
0.00	220.00	106.48	67.90	61.53	0.35	1.10	1.57
220.00	440.00	837.30	389.01	220.00	2.79	1.77	2.15
440.00	660.00	1039.55	442.67	220.00	3.46	2.01	2.35
660.00	880.00	1408.83	531.55	220.00	4.70	2.42	2.65
880.00	1100.00	2148.93	684.80	220.00	7.16	3.11	3.14
LB 1100.00	1120.00	384.83	79.28	20.01	1.28	3.96	4.85
1120.00	1140.00	488.28	91.46	20.01	1.63	4.57	5.34
1140.00	1160.00	601.34	103.63	20.01	2.00	5.18	5.80
1160.00	1180.00	723.62	115.81	20.01	2.41	5.79	6.25
1180.00	1200.00	854.80	127.98	20.01	2.85	6.40	6.68
1200.00	1220.00	992.66	139.99	20.01	3.31	7.00	7.09
1220.00	1240.00	1093.49	148.34	20.00	3.64	7.42	7.37
1240.00	1260.00	1191.42	156.19	20.01	3.97	7.81	7.63

Error / Warning and Notes

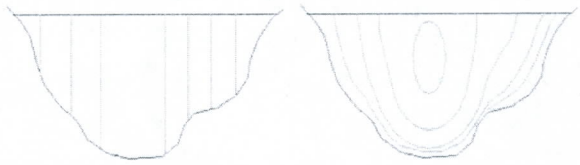
The energy loss was greater than 1.0 ft (0.3 m) between the current and previous cross section. That may indicate the need for additional cross sections.

32

Cautions in Using the Flow Distribution Option

WEST CONSULTANTS, INC.
www.westconsultants.com

- Multi-Dimensional Reality of Flow
  - In general, the results of the flow distribution computations should be used cautiously.
  - The results based on 1-D computations.
  - To achieve such detail, the user would need to use a three-dimensional hydraulic model, or measure the flow distribution in the field.



HEC-RAS Velocity Distribution

Representation of True Velocity Distribution

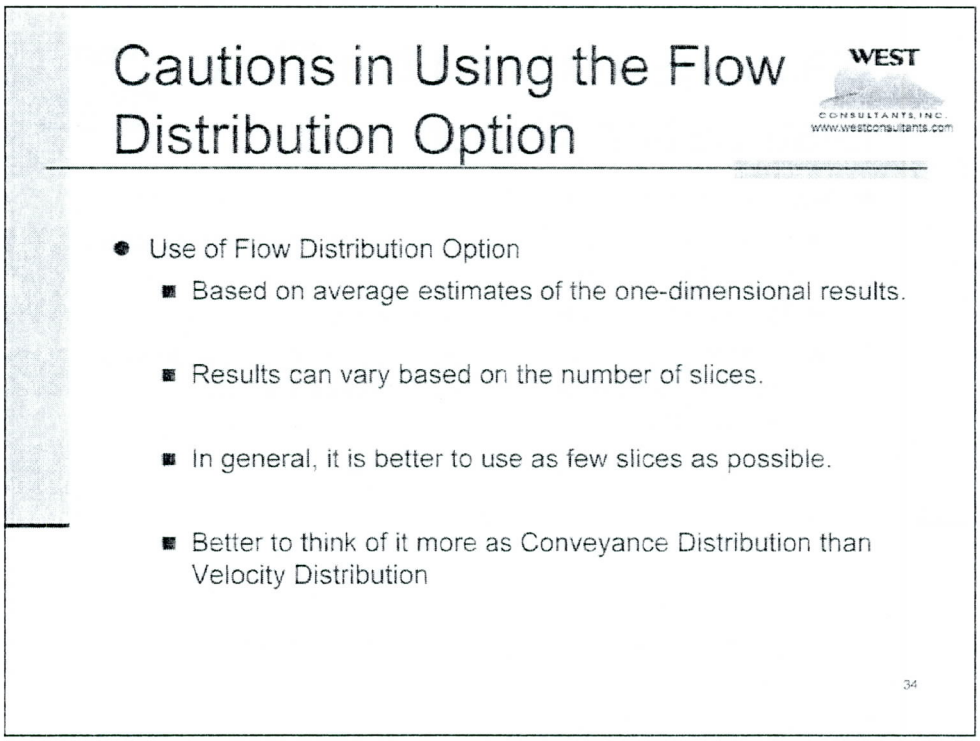
33

Multi-Dimensional Reality of Flow

In general, the results of the flow distribution computations should be used cautiously.

The results shown are based upon one-dimensional computations and true velocity and flow distribution vary vertically as well as horizontally.

To achieve such detail, the user would need to use a three-dimensional hydraulic model, or measure the flow distribution in the field.



Use of Flow Distribution Option

While the results from the flow distribution provided by HEC-RAS are better than the standard three subdivisions (left overbank, main channel, and right overbank) provided by the model, the values are still based on average estimates of the one-dimensional results.

The results obtained from the flow distribution option can vary with the number of slices used for the computation.

In general, it is better to use as few slices as possible.

The slices that are created are more representative of conveyance slices over velocity slices.

Performing a Water Quality Analysis

WEST CONSULTANTS, INC. www.westconsultants.com

- One-dimensional advection-dispersion equation
- Fate and transport of:
  - Water temperature
  - Arbitrary conservative and non-conservative constituents
  - Dissolved nitrogen
  - Dissolved phosphorus
  - Algae
  - CBOD
  - Dissolved oxygen
- Must have a working, calibrated model first

35

The Water quality module uses the QUICKEST-ULTIMATE explicit numerical scheme to solve the one-dimensional advection-dispersion equation. Individual sources and sinks as well as computed concentrations are available model output. The model simulates fate and transport of water temperature, arbitrary conservative and non-conservative constituents, dissolved nitrogen, dissolved phosphorus, algae, CBOD, and dissolved oxygen. In order to run the water quality model, a working, calibrated HEC-RAS unsteady or steady flow model must already be in place. (HEC)

Water Quality Data Entry

WEST CONSULTANTS, INC. www.westconsultants.com

- Water quality constituents
  - Temperature modeling
  - Nutrient modeling
    - Requires temperature modeling
  - Arbitrary constituent modeling
    - Independent of temperature and nutrient modeling
- Water quality data
  - Changes based on modeling technique used
  - Boundary conditions
  - Initial conditions
  - Observed data

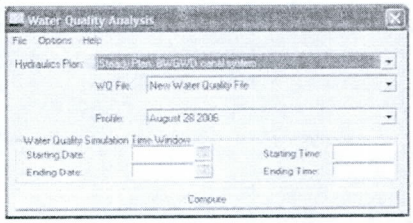
36

The model organizes constituents and sources and sinks into three major groups. Temperature modeling computes heat energy sources and sinks and water temperature. Nutrient modeling simulates nutrients, dissolved oxygen, CBOD, and algae. Because most of the rate constants in the nutrient model are temperature dependent, nutrients may not be modeled unless water temperature is also simulated. Arbitrary constituents are simple tracers, configured by the user. Arbitrary constituents are independent of water temperature and nutrients. (HEC)

Water Quality Analysis

WEST CONSULTANTS, INC.
www.westconsultants.com

- Run steady flow model
- Enter necessary water quality parameters
- Click on the Water Quality Analysis button



37

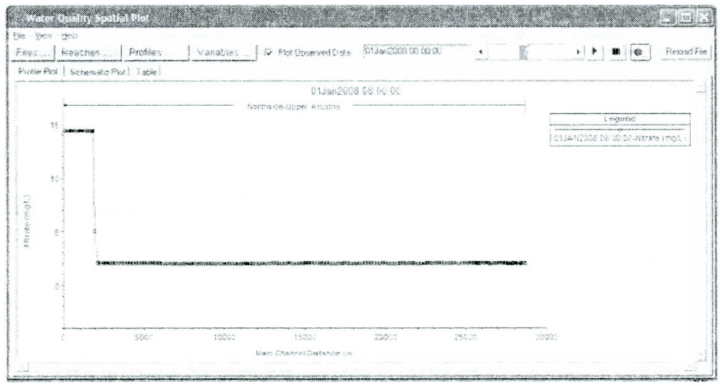
Water Quality Analysis Window:

1. Select appropriate hydraulic plan
2. Select appropriate water quality data file
3. Select appropriate steady flow profile to be used in the analysis
4. Enter a starting and ending time information (not constrained by hydraulic model because a steady model is used)
5. Hit the compute button

Viewing Water Quality Results

WEST
CONSULTANTS, INC.
www.westconsultants.com

- Water quality spatial plots

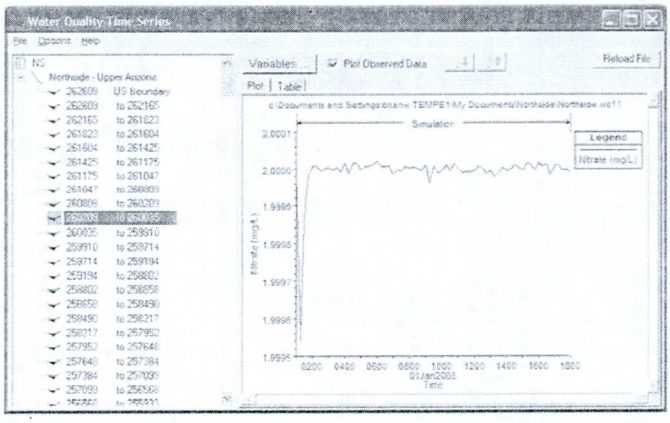


Spatial results along the channel

Viewing Water Quality Results

WEST CONSULTANTS, INC.
www.westconsultants.com

- Water time series plots



Time series of results at a particular location.

WEST

CONSULTANTS, INC.
www.westconsultants.com

Introduction to Unsteady Flow Modeling with HEC-RAS

- **Objective**: This lecture will show you the basics of creating and running a RAS unsteady flow model.

This lecture was originally developed by the staff and engineers at the US Army Corps of Engineers Hydrologic Engineering Center (HEC).

1

Topics to Discuss

WEST CONSULTANTS, INC.
www.westconsultants.com

- Overview
- Geometric data
- Unsteady flow data
  - Boundary and initial conditions
- Hydraulic property tables
- Unsteady flow simulation manager
  - Creating an unsteady simulation
- Post-processor
- Additional graphics/tables to view results

2

Overview

WEST CONSULTANTS, INC.
www.westconsultants.com

- Same geometry for steady and unsteady flow
- Simple dendritic streams to complex networks can be modeled
- Same hydraulic structures for steady and unsteady flow
- Capable of modeling a wide variety of hydraulic structures (e.g., bridges, culverts, levees, etc.)
- Uses the UNET equation solver (Dr. Robert Barkau)
  - Relatively fast matrix solver

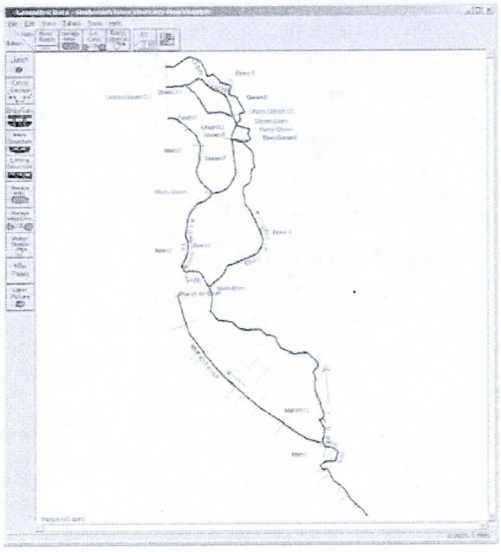
3

A key element in the design and development of HEC-RAS was to ensure that the same hydraulic calculation routines were used for both steady and unsteady flow computations (HEC). HEC reports that the matrix solver is relatively fast when compared to FLDWAVE and DWOPER (Apr. 2001 course notes). RUNUNET, a Windows interface, is available to run UNET from Windows 95, 98 or NT. See RAS User's Manual, April 2001, Chapter 10.2 (WEST 05.02.02)

Geometric Data Editor

WEST CONSULTANTS, INC.
www.westconsultants.com

- Geometry is entered in the same way for steady or unsteady flow models
- Additional data may be required



Snohomish River, Washington

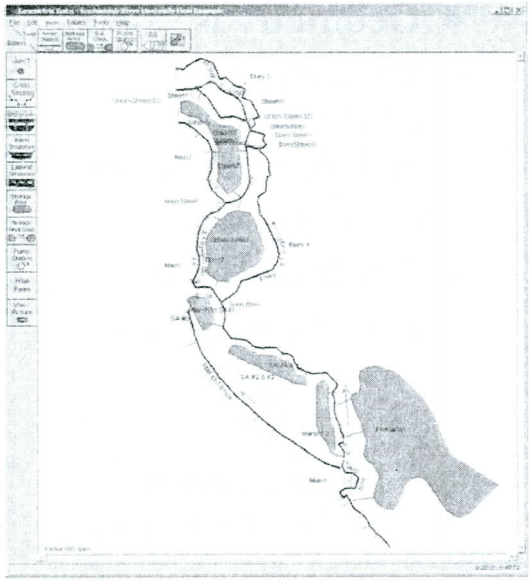
4

The same Geometric Data files can essentially be used interchangeably between a steady flow and unsteady flow model. For example, storage areas can be used in a steady flow model and will have a constant computed water surface elevation. However, the user needs to have full understanding of how to construct an unsteady flow model and the implications of the different modeling methods (e.g., permanent vs. non-permanent ineffective flow areas) (WEST 05.02.02). Additional data may also be required (e.g., HTab data) (WEST 07.09.03).

Optional Geometric Features

WEST CONSULTANTS, INC.
www.westconsultants.com

- Storage areas
- Storage area connections
- Inline and lateral structures
- Time or elevation controlled gates
- Bridges and culverts
- Pumps



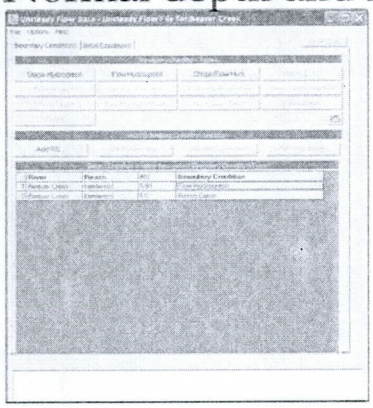
5

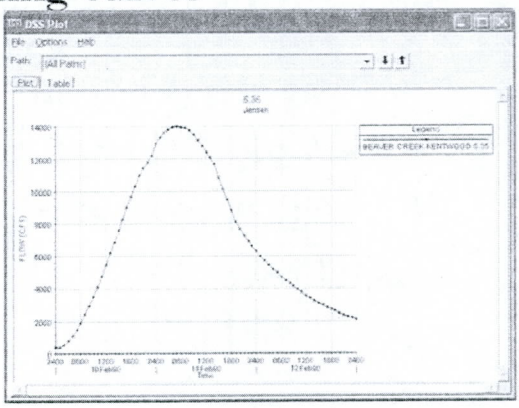
All of the existing hydraulic analysis features in the previous steady flow versions of HEC-RAS work within the new unsteady flow computations. The following new features were added to work with unsteady flow, but they also work in the steady flow simulation: lateral structures; storage areas (used to model areas of ponded water); storage area connections (used to exchange water between storage areas, or between a storage area and a river reach) (HEC).

Unsteady Flow Editor – Boundary Conditions

WEST CONSULTANTS, INC.
www.westconsultants.com

- Flow and stage hydrographs
  - Data can be entered manually or specified from a DSS file
- Normal depth and rating curves





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. Flow and stage data can be entered manually in a quick and efficient manner within the Unsteady Flow Data Editor. Data can be copied and pasted to and from spreadsheets such as Microsoft Excel, or interpolated between specified values. The specifics of each option are discussed in detail in the RAS User's Guide, January 2001, and covered in a later lecture (WEST 05.02.02).

Internal Boundary Conditions

WEST CONSULTANTS, INC.
www.westconsultants.com

- Lateral inflow to a node
- Uniform lateral inflow along a reach
- Groundwater interflow
  - Head specified over time and independent of river stage
- Time series of gate openings
- Elevation controlled gate
- Navigation dams
- Internal observed internal stage and/or flow hydrograph

7

Options for external boundary conditions are: stage hydrograph, flow hydrograph, combined stage and flow hydrograph (e.g., results from a forecast model where stage is observed until the beginning of the forecast time when flow is simulated), rating curve, or normal depth. The items in this figure are for internal boundary conditions.

Initial Conditions

WEST CONSULTANTS, INC. www.westconsultants.com

- Initial conditions must be established for the entire system by either:
  - Specifying flows for each branch; RAS then calculates the stages through a steady state simulation
  - Read in flow and stage at every node from a previous run (i.e., a "restart" file)

8

Initial flows and stages are required to begin the unsteady flow calculations. One option is to use standard step backwater calculations are used to calculate the initial stages in the system from the user entered flows. Initial stages are set by the user for storage areas. In addition, a "restart" file can be used instead (this is often called a "hotstart" file). From the Unsteady Flow Analysis Editor, select **Options**, then **Output Options** to set the restart file options. Restart files have the extension *.rst (WEST 05.02.02).

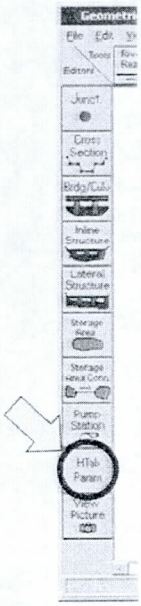
Unsteady Flow Simulation Procedure

WEST CONSULTANTS, INC.
www.westconsultants.com

- Geometric pre-processor
  - Processes geometric data into a series of hydraulic property tables
- Unsteady flow simulation
  - Performed using a modified version of UNET
- Post-processor
  - Computes detailed hydraulic information

Geometric Pre-processor

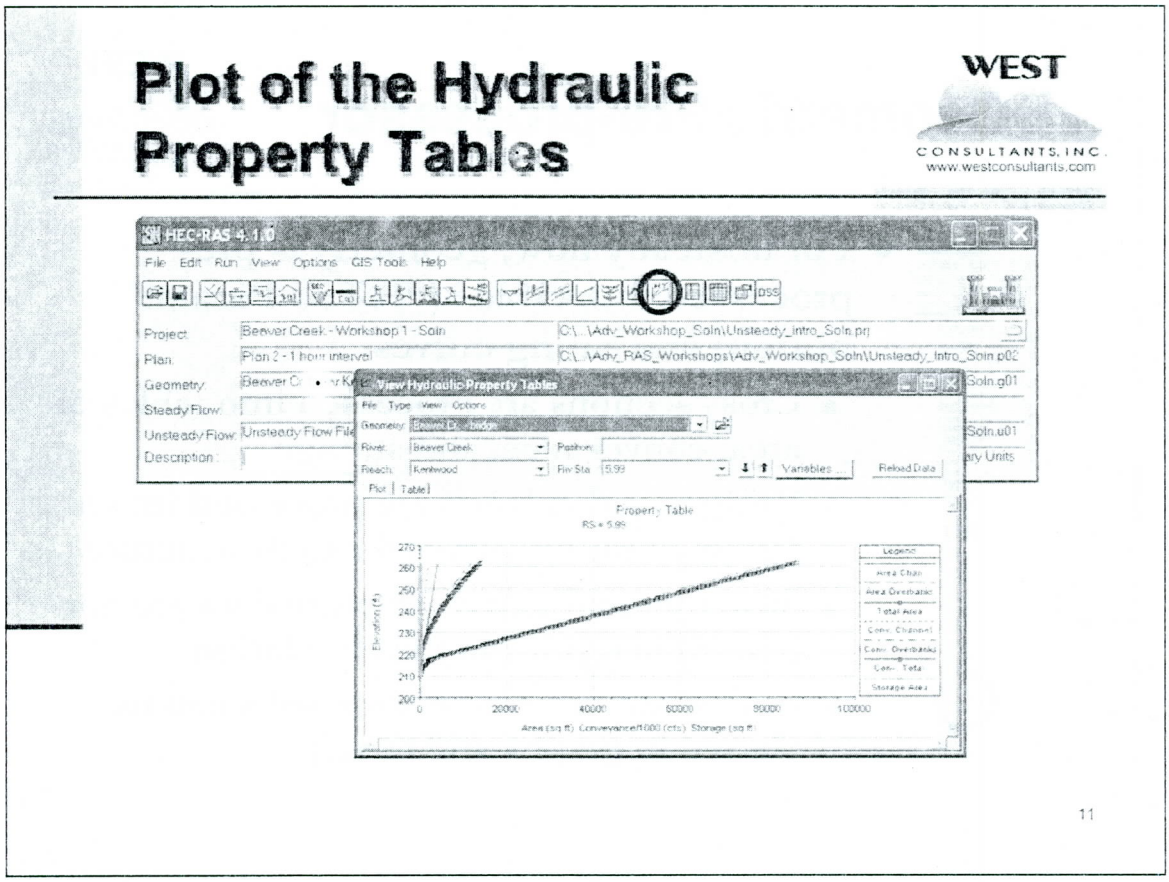
WEST CONSULTANTS, INC. www.westconsultants.com



- For unsteady flow, geometry is pre-processed into tables ("HTab Param." button) and rating curves
  - Cross sections are processed into tables of area, conveyance, and storage
  - Bridges and culverts are processed into a family of rating curves for each structure
  - Structure and gated structure data are not determined prior to the simulation
  - Geometric pre-processor results can be viewed in graphs and tables

10

The geometric pre-processor is used to process the geometric data into a series of hydraulic properties tables, rating curves, and families of 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 pre-processor must be executed at least once, but then only needs to be re-executed if something in the geometric data has changed** (HEC). The "HTab Param." button in the Geometric Data Editor shown in this slide is used for editing cross section hydraulic table parameters. The starting elevation, increment, and number of points are specified in the cross section HTab Editor. There is a similar button in the Bridge/Culvert Data Editor. Structures and gates could have an endless number of curves based on the variation of upstream and downstream head as well as various gate openings. Therefore structure and gated structure hydraulic tables are not determined in the geometric pre-processor. See the next slide for discussion on the hydraulic table parameters and plots (WEST 06.07.02).



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 (HEC).

Hydraulic Property Tables

WEST CONSULTANTS, INC.
www.westconsultants.com

View Hydraulic Property Tables

File Type

Geometry: Beaver Cr. - bridge

River: Beaver Creek

Reach: Kentwood Riv Sta: 5.99 Variables ... Reload Data

Plot Table

Minimum Elevation: 209.9 Chan Length: 100 Wtd Overbank Length: 70 Preiss WD: 0 Preiss Elev:

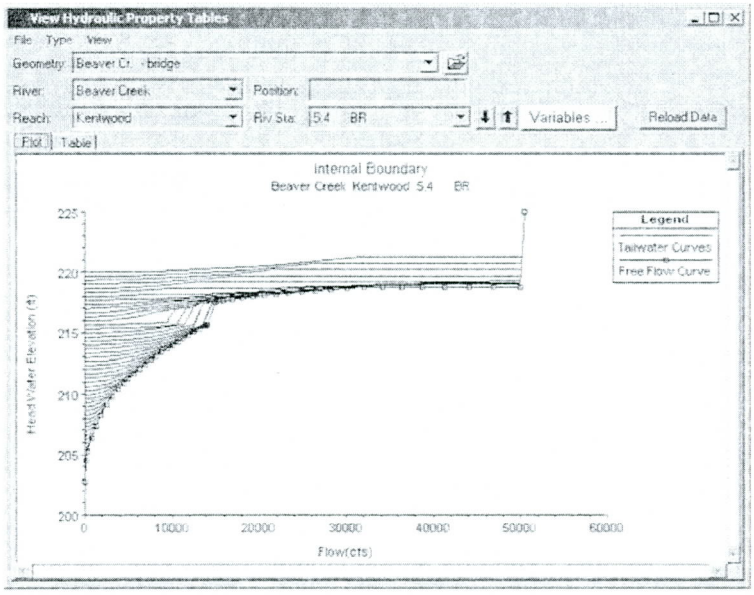
	Elevation (ft)	Area Chan (sq ft)	Area L+R (sq ft)	Area Total (sq ft)	Storage Area (sq ft)	Conv Ch (cfs)	Conv L+R (cfs)	Conv Total (cfs)	Top Width (ft)	Alpha
1	210.90	30.53	150.59	181.12	0.00	958	2832	3790	114.75	1.17
2	211.41	51.46	193.49	244.94	0.00	2141	3627	5768	135.54	1.56
3	211.92	74.37	246.29	320.66	0.00	3728	4689	8417	164.42	1.91
4	212.43	99.27	313.84	413.11	0.00	5712	6082	11794	198.12	2.20
5	212.94	126.16	396.55	522.71	0.00	8097	7577	15674	232.64	2.56
6	213.45	155.04	503.38	658.42	0.00	10894	8787	19681	308.12	3.21
7	213.96	185.91	639.74	825.65	0.00	14111	11687	25798	342.62	3.38
8	214.47	219.99	787.86	1007.85	0.00	17075	15779	32853	372.31	3.13
9	214.98	258.64	946.28	1204.92	0.00	21089	20605	41694	396.93	3.00
10	215.49	298.80	1112.25	1411.05	0.00	26531	26256	52787	411.44	3.03
11	216.00	339.58	1285.01	1624.58	0.00	32484	32544	65027	425.96	3.05
12	216.51	380.97	1507.85	1888.83	0.00	38933	40416	79349	634.71	3.11
13	217.02	422.78	1896.88	2319.67	0.00	46225	51072	97298	932.93	3.45
14	217.53	464.60	2337.33	2801.94	0.00	54094	65400	119494	958.35	3.61
15	218.04	506.42	2790.75	3297.17	0.00	62451	82579	145029	983.76	3.65
16	218.55	548.24	3450.21	3998.45	0.00	71280	103249	174530	1643.00	3.94
17	219.06	590.06	4290.46	4880.52	0.00	80571	129303	209874	1810.23	4.24
18	219.57	631.88	5182.63	5814.51	0.00	90311	160897	251208	1843.29	4.35
19	220.08	673.70	6086.75	6760.45	0.00	100491	197169	297660	1866.32	4.33
20	220.59	715.52	7003.09	7718.60	0.00	111101	227541	338642	1892.80	4.25

12

Clicking the "Table" tab will show the hydraulic property table data (WEST 06.07.02).

Bridge Hydraulic Properties Plot

WEST CONSULTANTS, INC.
www.westconsultants.com



13

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 (HEC).

Creating an Unsteady Simulation

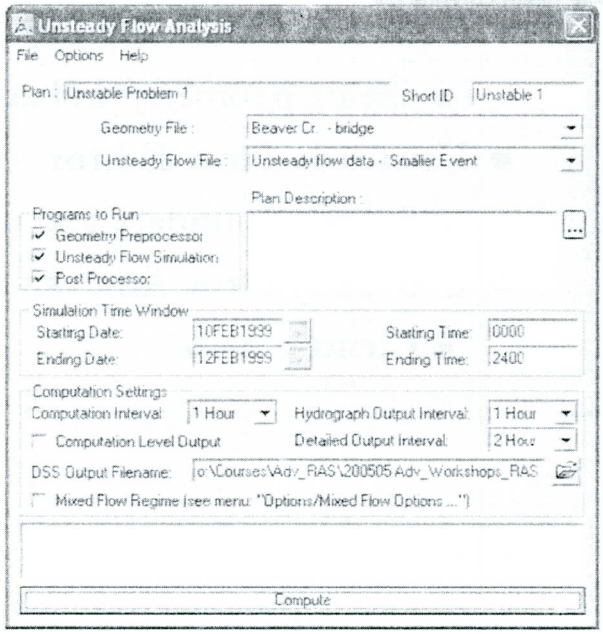
WEST CONSULTANTS, INC.
www.westconsultants.com

- Geometric Data Editor
  - Create geometry and add structures
- Unsteady Flow Editor
  - Specify boundary and initial conditions
- Unsteady Flow Analysis Editor
  - Create a plan

Unsteady Flow Analysis Editor

WEST CONSULTANTS, INC.
www.westconsultants.com

1. Define a Plan
2. Select which programs to run
3. Enter a starting and ending date and time
4. Set the computation settings
5. Press the Compute button



Once all of the geometric 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 (a tool button is also available). The unsteady flow computations within HEC-RAS are performed by a modified version of the UNET (Unsteady NETwork model) program, developed by Dr. Robert Barkau (Barkau, 1992). The unsteady flow simulation is actually a three step process. First, a program reads data from a HEC-DSS file and converts it into the user specified computation interval. Next, the UNET program runs. UNET reads the hydraulic properties tables computed by the geometric pre-processor, as well as the boundary conditions and flow data. The program (UNET) then performs the unsteady flow calculations. The final step takes the results from the UNET unsteady flow run and writes them to a HEC-DSS file (HEC).

Barkau, R.L. (1992). *UNET, one-dimensional unsteady flow through a full network of open channels*, computer program, St. Louis, MO.

Simulation Options

WEST CONSULTANTS, INC. www.westconsultants.com

- Stage and flow output locations
- Flow distribution locations
- Flow roughness change factors
- Seasonal roughness change factors
- Additional output options
  - Write restart file
  - Debugging level output file
- Calculation options and tolerances

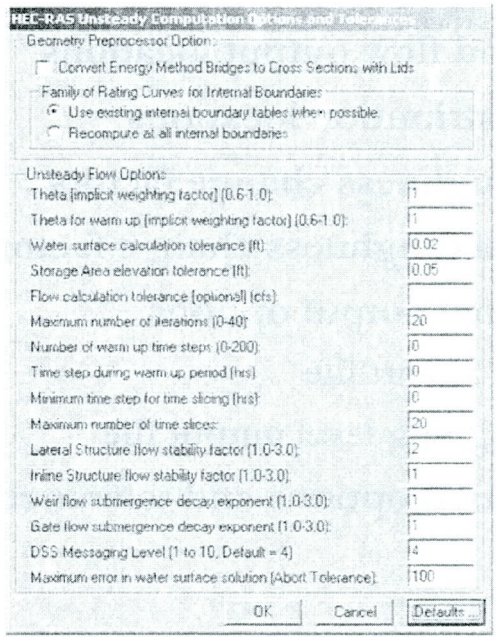
16

These options are all accessed from the Unsteady Flow Analysis Editor, under **Options** (WEST 05.02.02). Default locations for stage and flow output hydrographs are the upstream and downstream ends of every reach and at bounding sections of structures (e.g., bridges and structures). Flow roughness change factors are set in calibration (April 2001 course notes).

Additional simulation options include unsteady encroachments, dam (inline structure) breach, levee (lateral structure) breach, and mixed flow options. These options are discussed in a later lecture (WEST 12.19.03).

Calculation Options and Tolerances

WEST CONSULTANTS, INC.
www.westconsultants.com



17

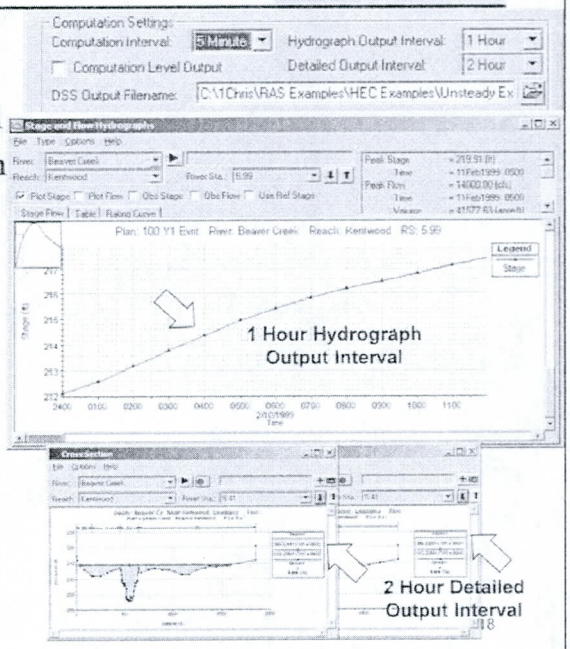
This editor is used to set a number of options and tolerances for unsteady flow modeling, which are discussed in Chapter 8 of the RAS User's Manual, as well as in detail in later lectures. Briefly, theta is a stability factor, which can be set to 1 (most stable, least accurate) for initial simulations and then preferably lowered towards 0.6 (least stable, most accurate) for final simulations. Computation tolerances are set in this editor (you may want to temporarily increase the tolerance values if you are having trouble successfully completing a simulation, and then return to the lower values once the stability issues have been corrected) as well as factors affecting structure and spillway computations (WEST 06.07.02). Structure decay exponents are used to stabilize highly submerged structures and should range between 1 and 3 (personal communication with HEC on 05.30.02).

Convert Energy Method Bridges to Cross Sections with Lids: This option is used to convert bridges to normal cross sections instead of being processed as a family of rating curves. If you have a bridge in which you are using the energy solution method for high and low flow solutions, there is no need to process this as a family of rating curves. Instead, you can have the program treat the two internal bridge cross sections as any other normal cross section. If you turn this option on, the program will create a separate table of elevation versus area and conveyance for each of the two internal bridge sections (HEC).

Computational Settings

WEST CONSULTANTS, INC.
www.westconsultants.com

- The computation interval must be less than or equal to any boundary condition intervals
- Hydrograph output interval defines the spacing of points on the hydrograph
- Detailed output interval defines the time steps when animating



The computation time interval must be equal to or less than the time step manually entered at the boundary condition. For example, the computation interval must be 15 minutes or less, if flow data at an upstream boundary are entered at 15 minute intervals. If this rule is broken, the warning message "Boundary condition interval is less than the computation interval" appears in the Unsteady Flow Analysis Editor (HEC).

Post-Processor (Detailed Output)

WEST CONSULTANTS, INC. www.westconsultants.com

- Used to compute detailed hydraulic information for a set of user-specified time lines and an overall maximum water surface profile
- Computed stages and flows are passed to the steady flow program for the computation of detailed hydraulic results

19

The post-processor is used to compute detailed hydraulic information for a set of user specified time lines during the unsteady flow simulation period. In general, the UNET program only computes stage and flow hydrographs at user specified locations. **If the post-processor is not run, then the user will only be able to view the stage and flow hydrographs and no other output from HEC-RAS**. By running the post-processor, the user will have all of the available plots and tables for unsteady flow that HEC-RAS normally produces for steady flow. When the post-processor runs, the program reads from HEC-DSS the maximum water surface profile (stages and flows) and the instantaneous profiles. These computed stages and flow are sent to the HEC-RAS steady flow computation program SNET. Because the stages are already computed, the SNET program does not need to calculate a stage, but it does calculate all of the hydraulic variables that are normally computed. This consists of over two hundred hydraulic variables that are computed at each cross section for each flow and stage (HEC).

Viewing Unsteady Flow Results

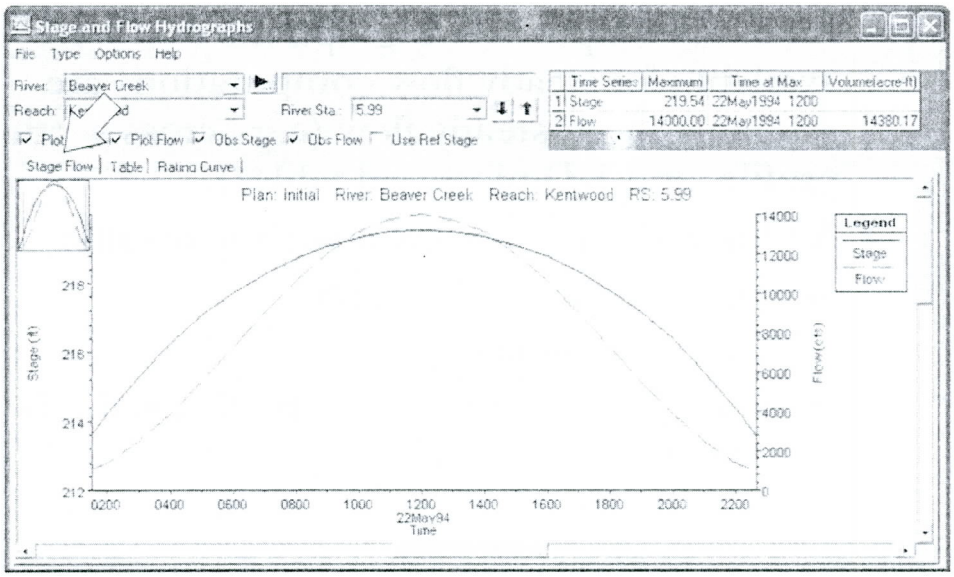
WEST CONSULTANTS, INC.
www.westconsultants.com

- All of the output viewing options that were available for steady flow computations are available for unsteady flow (e.g., cross section, profile, and 3-D plots and tables).
- Additional ways to view unsteady results:
  - Stage and flow hydrographs
  - Time series tables
  - Animation of cross section, profile, and 3-D graphics

20

Stage and Flow Hydrographs

WEST CONSULTANTS, INC.
www.westconsultants.com

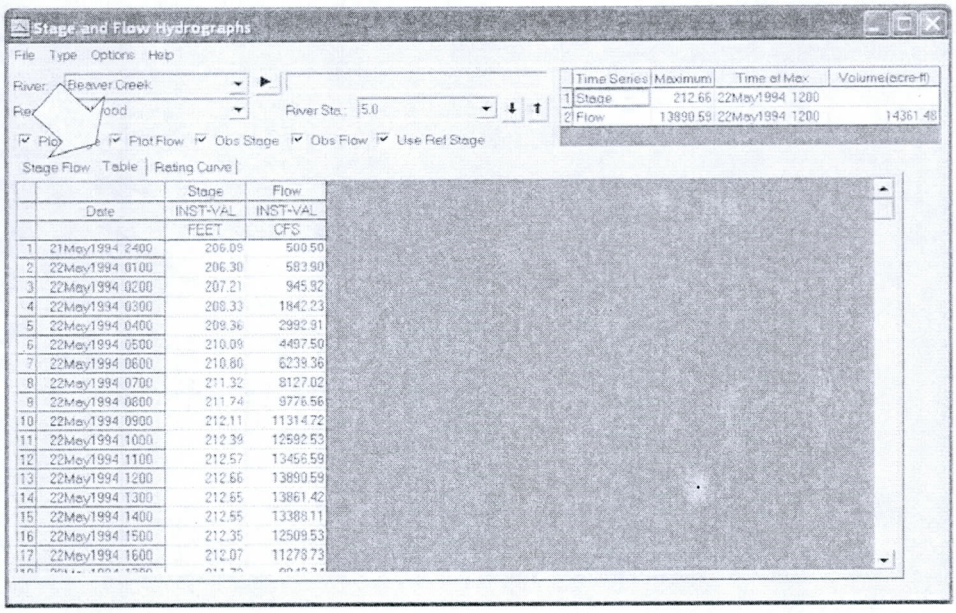


21

The stage and flow hydrograph button allows the user to plot flow hydrographs, stage hydrographs, or both simultaneously. Additionally, if the user has observed hydrograph data, those data can also be plotted at the same time. The plot can be printed or copied to the Windows clipboard for use in other software programs (HEC).

Time Series Table

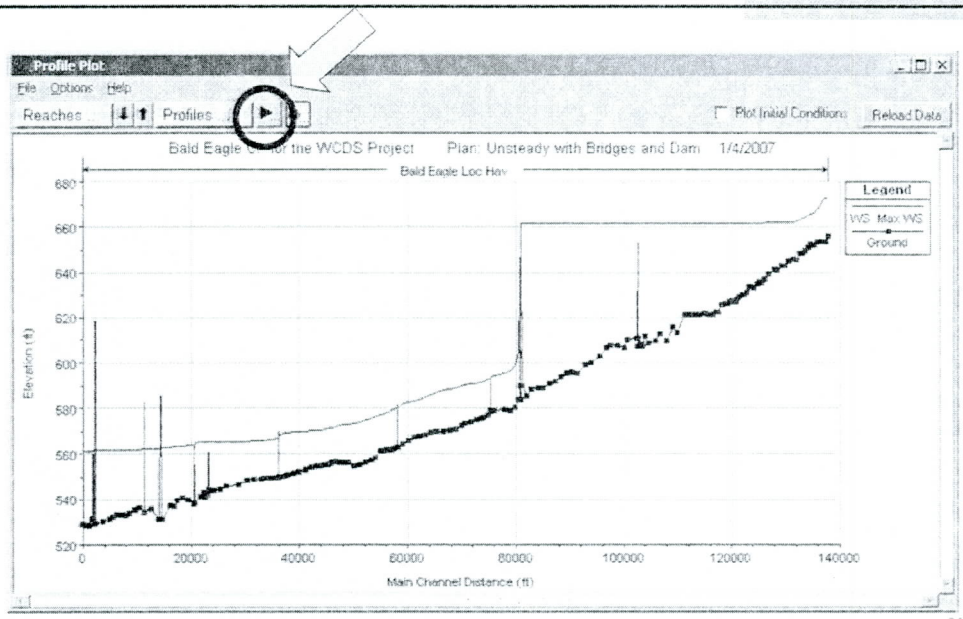
WEST CONSULTANTS, INC.
www.westconsultants.com



By simply pressing the "Table" tab from the Flow and Stage Hydrographs plot, the user will get a table containing the data in a time series format. These data can be printed or copied to the Windows clipboard for use in other software programs (HEC).

Animation

WEST
CONSULTANTS, INC.
www.westconsultants.com



23

The HEC-RAS software has the ability to animate the cross section, profile, and 3-D graphics. When the user selects the "Animate" button, HEC-RAS steps through the computed results in a timed sequence (HEC). The Animation Control Editor (accessed under **Options** from cross section, profile, or 3-D plots or by selecting the "Animate" button) allows the user to control the speed of the animation by clicking on the "…" button and then selecting the delay (WEST 06.07.02).

WEST

CONSULTANTS, INC
www.westconsultants.com

Detailed Output and Results for Unsteady Flow Modeling

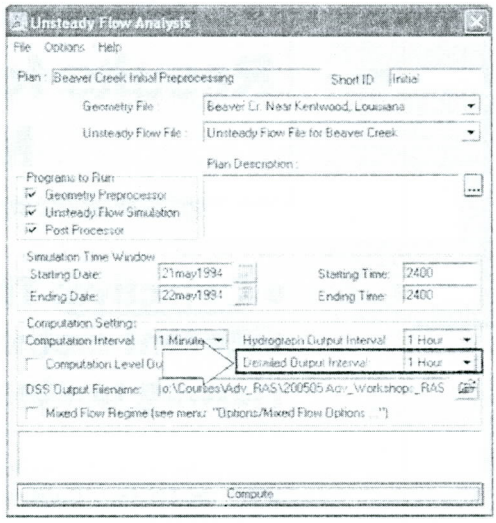
- Objective: This lecture will present different methods for generating and viewing unsteady flow results in RAS.

This lecture was originally developed by the staff and engineers at the US Army Corps of Engineers Hydrologic Engineering Center (HEC).

Output from Programs

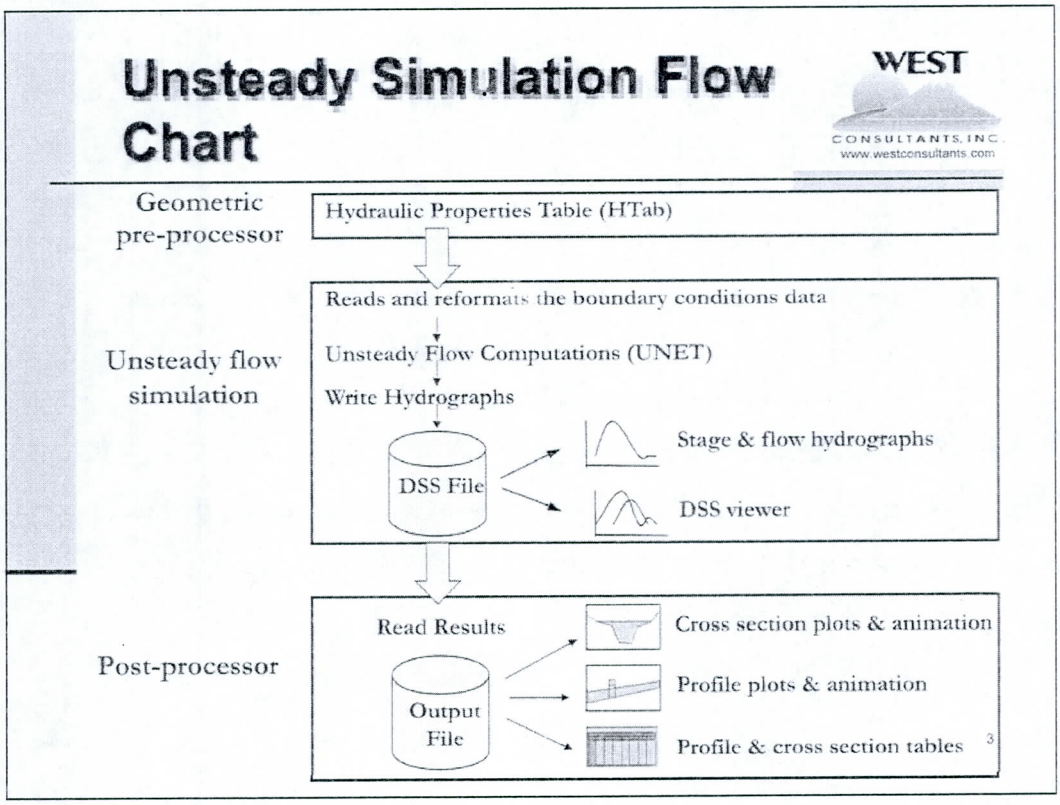
WEST CONSULTANTS, INC.
www.westconsultants.com

- Unsteady flow simulation
  - Stage and flow hydrographs
  - Log file output
    - Search for warning
- Post-processor
  - Detailed output
  - Animation data



2

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 for a series of snap shots in time based on the selected "Detailed Output Interval" (HEC).



The log file is a text file, and can be viewed with any text file viewer such as Notepad. DSS files are in a special binary format that requires a program specifically designed to work with it (e.g., HEC-RAS, HEC-DSSVue). The output files from the post-processor are in the same format as the output files from the steady flow computation engine and require RAS in order to be viewed (HEC).

Locations for Writing Hydrographs

WEST CONSULTANTS, INC. www.westconsultants.com

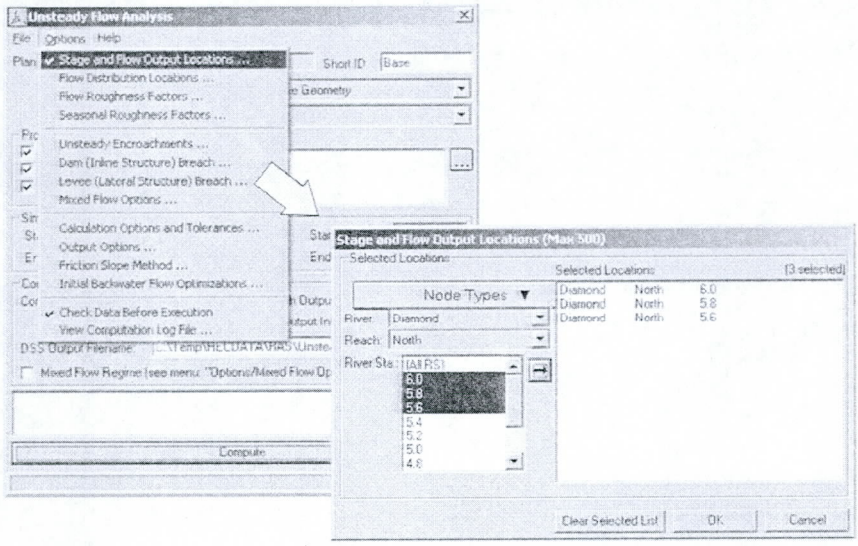
- Default locations
  - Upstream and downstream ends of all reaches
  - Upstream and downstream of internal boundaries (e.g., bridges and structures)
- User selected locations
  - Additional locations can be specified by the user
  - Set through "Options" in the Unsteady Flow Analysis Editor
- Can be created at a maximum of 500 locations

4

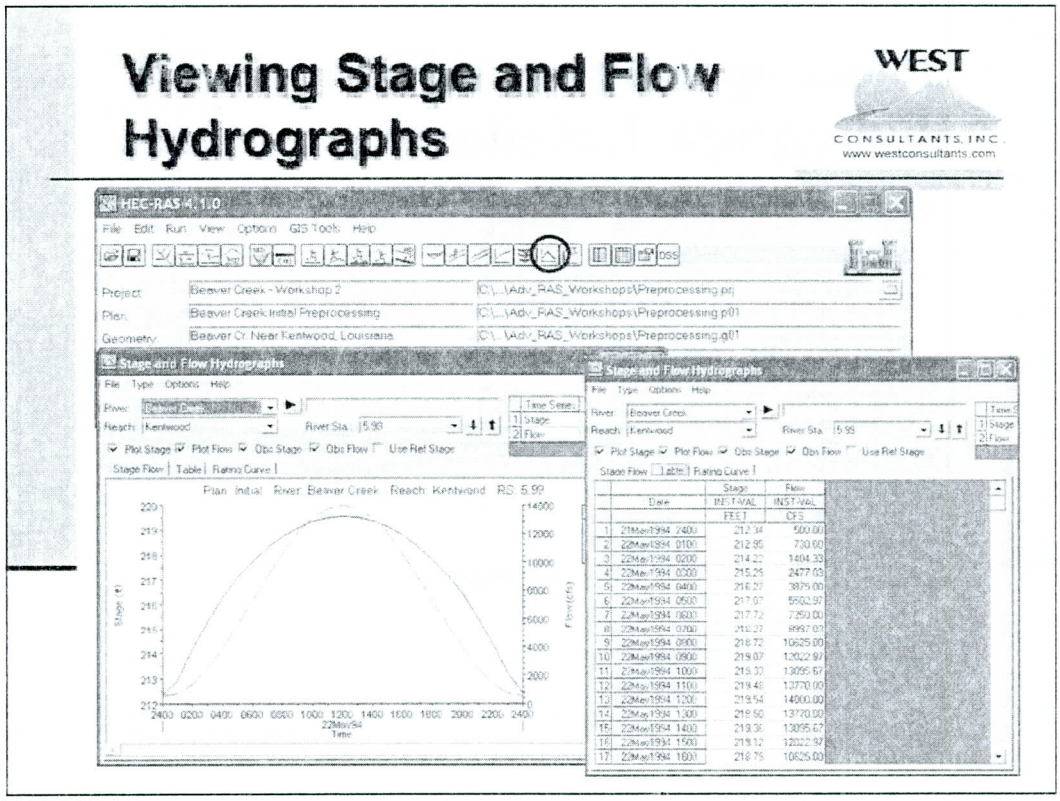
Stage and flow hydrographs are written by the unsteady computation engine at some default locations, but the user has the option to request hydrographs at desired locations (HEC).

Selecting Stage and Flow Hydrograph Locations

WEST CONSULTANTS, INC.
www.westconsultants.com



5

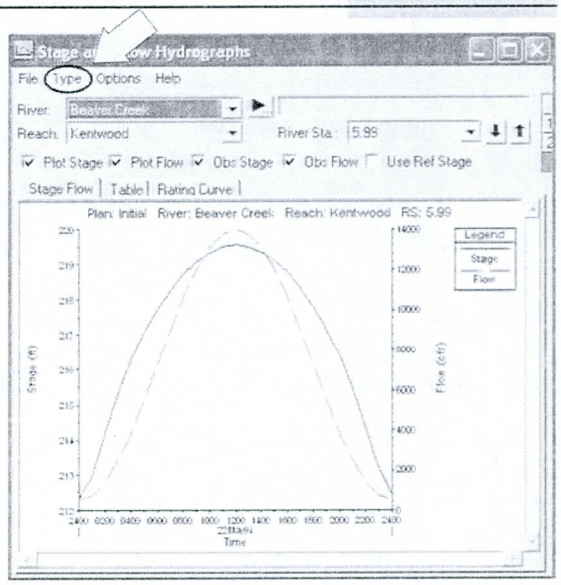


The Stage and Flow Hydrographs window is opened from the main RAS window. **Stage and Flow Hydrographs** can be selected from under the **View** menu. Alternatively, the button highlighted in the slide can be pressed to open the window. Data can be viewed graphically or in tabular form by selecting the desired tab. By default, both stage and flow are plotted. When there are both stage and flow on the plot, a second Y axis is displayed (HEC). Only simulated and observed stage and flow, at one particular cross section at a time, can be viewed in the Stage and Flow Hydrographs window. Other "types" of hydrographs (from **Type** on the menu bar) can be selected (see the next slide) (WEST 05.02.02).

"Types" of Stage and Flow Hydrographs

WEST CONSULTANTS, INC.
www.westconsultants.com

- Cross sections
- Bridges/culverts
- Inline structures
- Lateral structures
- Storage areas
- Storage area connections
- Pump stations
- Ground Water Interaction

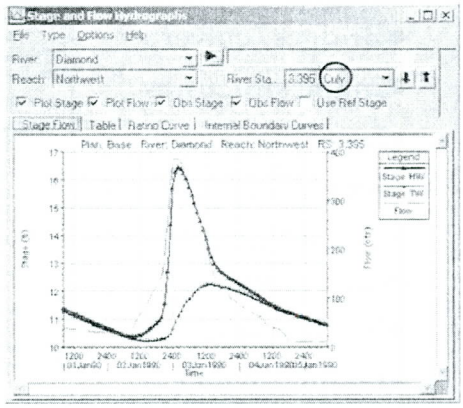


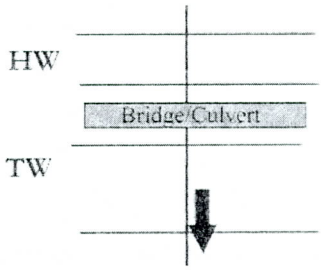
Hydrographs at various other "types" of locations can be viewed once the Stage and Flow Hydrograph window is opened (e.g., cross sections, bridges/culverts, inline structures, etc.). Again, only one structure can be viewed at a time (WEST 05.02.02).

Bridge and Culvert Hydrographs

WEST CONSULTANTS, INC
www.westconsultants.com

- Variables shown:
  - Stage at the upstream (headwater) and downstream (tailwater) bounding cross sections
  - Flow at the downstream cross section





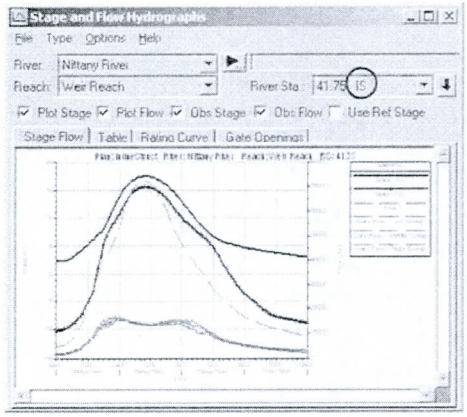
8

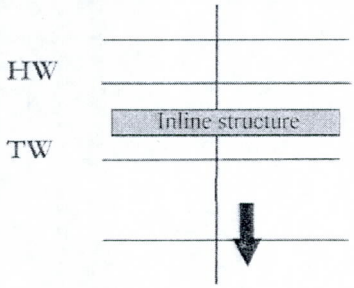
Stage plotted at the upstream and downstream cross sections of the bridge or culvert. Flow is plotted only at the downstream end of bridge or culvert (HEC).

Inline Structure Hydrographs

WEST CONSULTANTS, INC.
www.westconsultants.com

- Variables shown:
  - Stage (headwater)
  - Stage (tailwater)
  - Flow total (tailwater)
  - Flow in gates



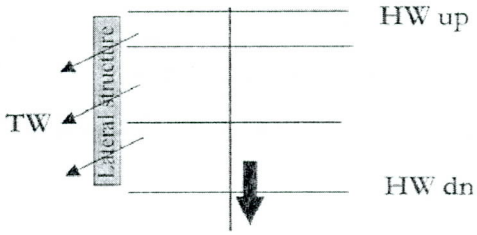


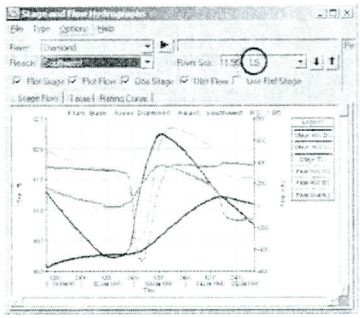
9

Lateral Structure Hydrographs

WEST CONSULTANTS, INC.
www.westconsultants.com

- Variables shown:
  - Stage and flow at upstream end of structure
  - Stage and flow at downstream end of structure
  - Stage at downstream end of structure
  - Flow leaving
  - Individual flow through each gate (if present)

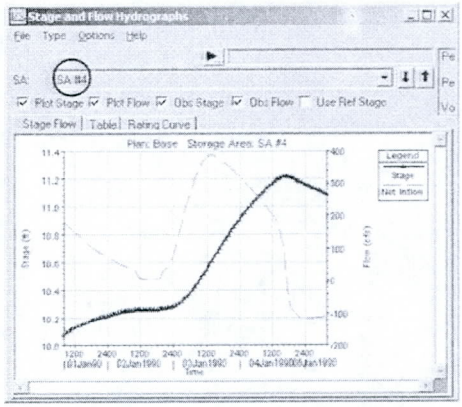


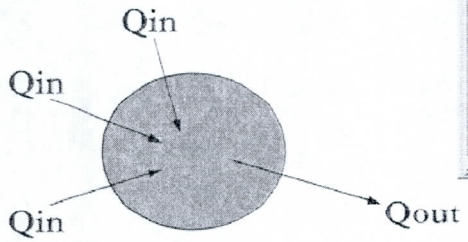


Storage Area Hydrographs

WEST CONSULTANTS, INC
www.westconsultants.com

- Variables shown:
  - Stage in storage area
  - Net inflow to storage area = Σ Qin - Σ Qout





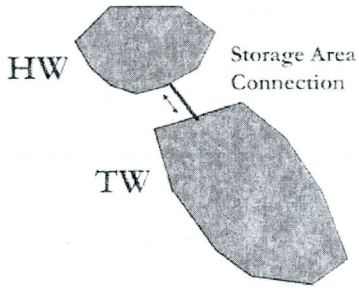
11

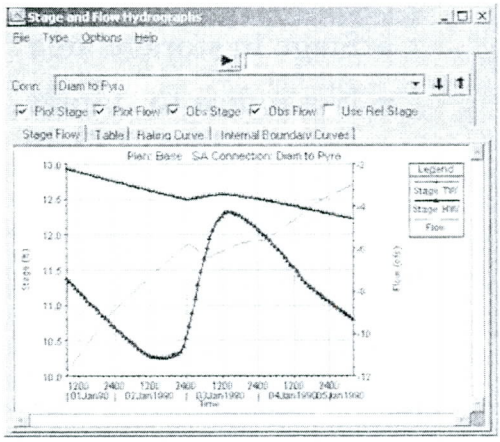
The only flow information saved for a storage area is stage and the net inflow. The net inflow is the sum of all flows into a storage area (flows leaving are considered negative). For information about the individual links to a storage area, go to the storage area connection or lateral structure specific output (HEC).

Storage Area Connection Hydrographs

WEST CONSULTANTS, INC.
www.westconsultants.com

- Variables shown:
  - Stage (headwater)
  - Stage (tailwater)
  - Flow





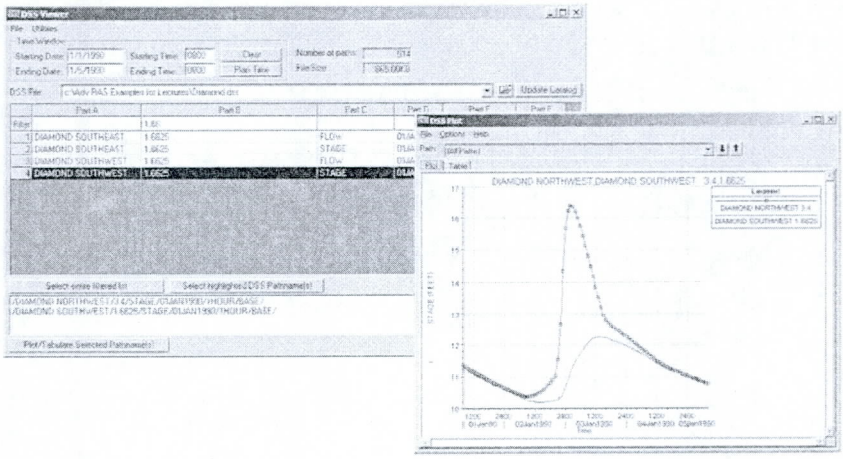
12

The output for a storage area connection consists of two stage hydrographs and a flow hydrograph (HEC).

Displaying Hydrographs Using the DSS Viewer

WEST CONSULTANTS, INC
www.westconsultants.com

- DSS Viewer can be used for user defined comparisons



13

The DSS Viewer in HEC-RAS can also be used to look at the same data shown in the Stage and Flow Hydrographs window. It is a little more laborious to find the desired paths, but in the DSS Viewer users have more options for comparisons. In the plot above, the inflow and outflow hydrographs for a long reach are compared (HEC). This is a good way to simultaneously compare hydrographs at two or more different locations (WEST 05.02.02). To get to the DSS Viewer, select **View** and then **DSS Data** from the main RAS window. Alternatively, the user can just press the "DSS" button on the main toolbar.

HEC-DSS Overview

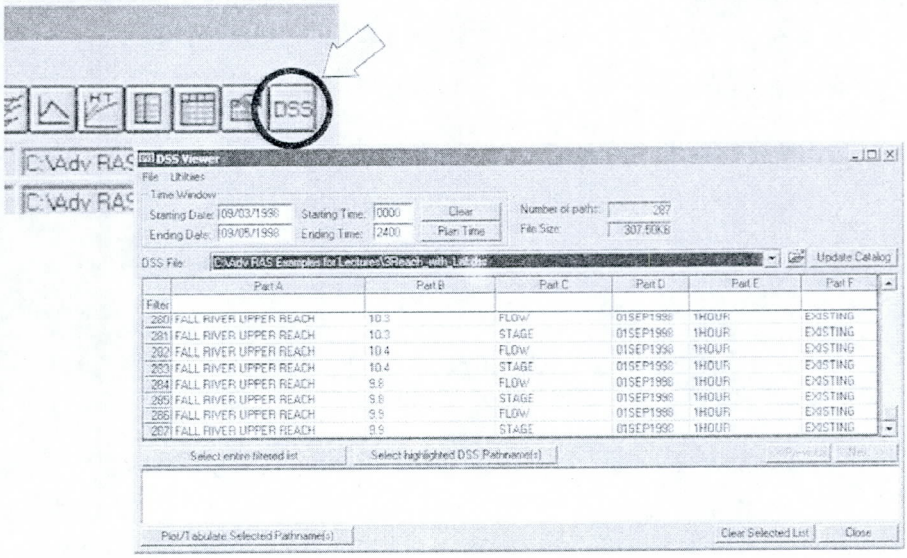
WEST CONSULTANTS, INC.
www.westconsultants.com

- Hydrologic Engineering Center's - Data Storage System
- Database that stores time series data such as stage and flow hydrographs
- Used with many HEC programs as a data source and storage for results
- Often used as data storage between applications

14

DSS Viewer in HEC-RAS

WEST CONSULTANTS, INC.
www.westconsultants.com



The DSS Viewer can be opened by clicking the "DSS" button (the last button on the right) on the main RAS window or by selecting the menu option **DSS Data** under **View**.

HEC-DSS Overview

WEST CONSULTANTS, INC. www.westconsultants.com

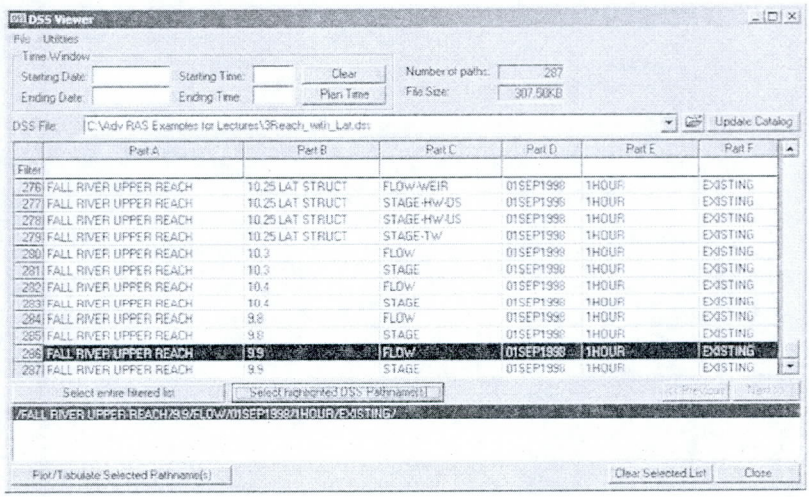
- Data are organized in a catalog of "paths"
- Paths have 6 parts (A-F)
  - Part A - River basin or project name
  - Part B - Location or gage identifier
  - Part C - Data variable (e.g., FLOW, STAGE)
  - Part D - Start for block of data
  - Part E - Time interval (e.g., 1DAY, 3HOUR, 5MIN)
  - Part F - Additional descriptor (e.g., "SIM" or "PLAN1")

16

The slide lists the typical uses for the six part names in DSS. In HEC-RAS, the F part is assigned the short plan ID from an unsteady simulation (WEST 06.07.02).

DSS Viewer - Selecting Paths

WEST CONSULTANTS, INC.
www.westconsultants.com

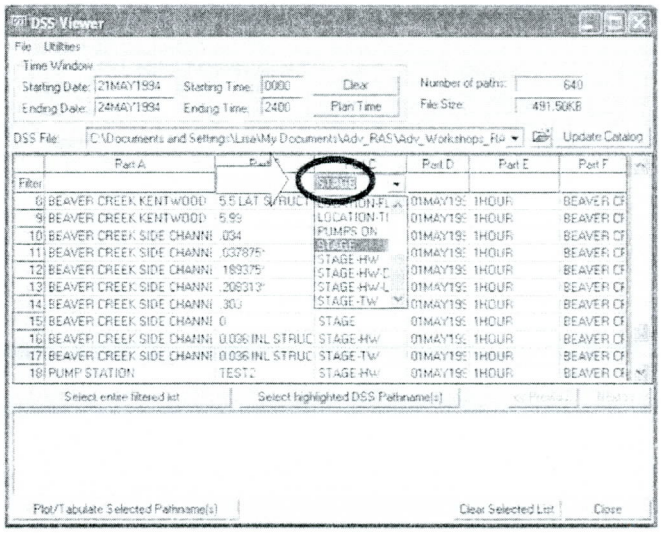


17

Select the DSS file using the browse button. Highlight the desired path and press the "Select Highlighted DSS Pathname(s)" button. The selected path then appears in the lower window. The button in the lower left says "Plot/Tabulate Selected Pathname(s)"; this option plots and tabulates the data. When viewing time series data the specific time window can be entered or left blank (this option will plot all of the data in a path). Select paths from the available list using standard Windows extended selection keys (i.e., using the Shift key to select a range, Ctrl key to pick individually, or Ctrl + A to select all). Plot the selected paths with the button on the lower left. A new window will open with the data plotted in it (HEC).

DSS Viewer - Filtering Paths

WEST CONSULTANTS, INC.
www.westconsultants.com

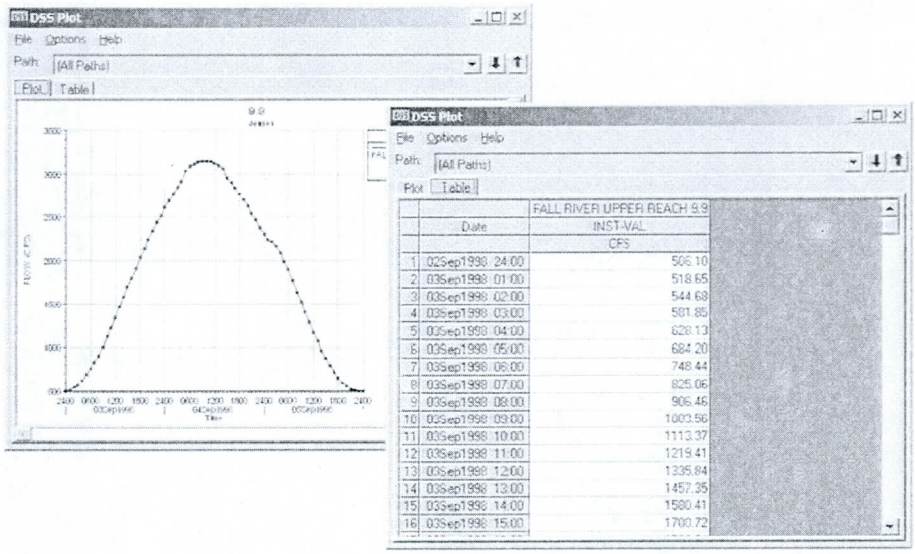


18

The path selection filter is used to narrow the list of paths displayed. This can greatly speed the search for a particular path or a certain subset of paths. The list filter is located on the first white line in the DSS Viewer labeled "Filter". To set a filter: click the mouse pointer in one of the filter boxes and a combination drop down list box will appear in the grid cell. The drop down list will contain a list of all the entries in this column. Select an entry in the drop down list or type some characters in. If the DSS file contained only STAGE and FLOW records, then the C Part filter list would contain only those two options. In the above example, the C Part is filtered to "STAGE", and only the DSS paths in the file that have "STAGE" as the first 5 characters are in the filtered list (HEC).

DSS Viewer - Plots and Tables

WEST CONSULTANTS, INC.
www.westconsultants.com



The DSS Plot window has two tabs, one for a graphical plot view and the other for a tabular view. All of the paths that were selected are in a drop down list labeled "Path". When the window first opens, the "Path" list box contains the text "(All Paths)". This will plot the all the selected paths in one plot. Alternatively, the user could select one path from the list and plot one at a time. The plot options are similar to the other plot options in RAS. The plot can be copied to the Windows clipboard or printed. The table can be copied to the Windows clipboard and transferred to an input table in RAS or to another program altogether (HEC).

Exporting HEC-RAS Results to DSS

WEST CONSULTANTS, INC. www.westconsultants.com

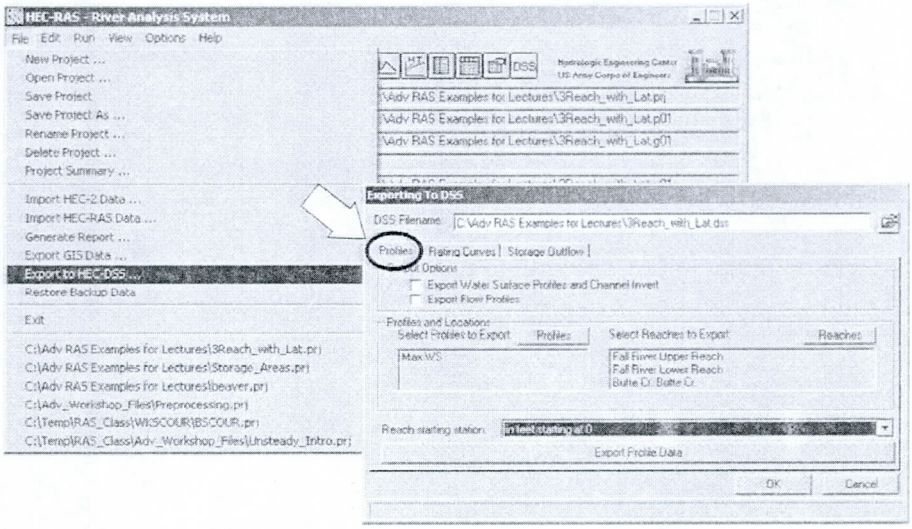
- Export to DSS:
  - Water surface profiles
  - Rating curves
  - Storage outflow information
- Both steady and unsteady results can be exported
- To export, select "File" and then "Export to HEC-DSS" from the main RAS window

20

Data can be exported from HEC-RAS to HEC-DSS, the types of data that can be exporter are: water surface profiles, rating curves, and storage outflow information. The export function gets data from either a steady flow output file or from an unsteady flow post-processor file. In general, when running unsteady flow computations, the profiles and rating curves will already be in a DSS file, but this utility is a good way of getting the family of rating curves at an internal boundary condition and the storage outflow data into DSS (HEC).

Exporting HEC-RAS Results to DSS – Water Surface Profiles

WEST CONSULTANTS, INC.
www.westconsultants.com



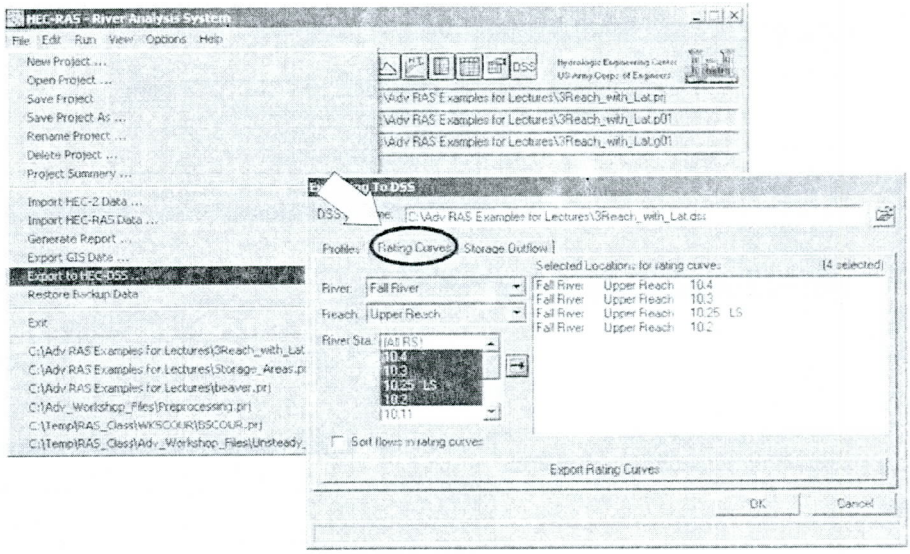
The first step in exporting any data to DSS is to select or name the DSS file. The file can be a new or an existing file. If the file already exists, the paths and data will be appended to the file. If the file already exists, any data with matching pathnames would be overwritten by the new data. The output options for profiles are water surface profiles (for plotting purposes the channel invert gets written as well) and flow profiles. Select the profiles and reaches to export with the "Profiles" and "Reaches" buttons. Select the river station starting position. There are 6 possibilities (similar choices for metric):

1. in feet starting at 0
2. in feet starting at lowest river station (assuming river station is in feet)
3. in feet starting at lowest river station (assuming river station is in miles)
4. in miles starting at 0
5. in miles starting at lowest river station (assuming river station is in feet)
6. in miles starting at lowest river station (assuming river station is in miles)

Press the "Export Profile Data" button to export the profiles to the DSS file (HEC).

Exporting HEC-RAS Results to DSS – Rating Curves

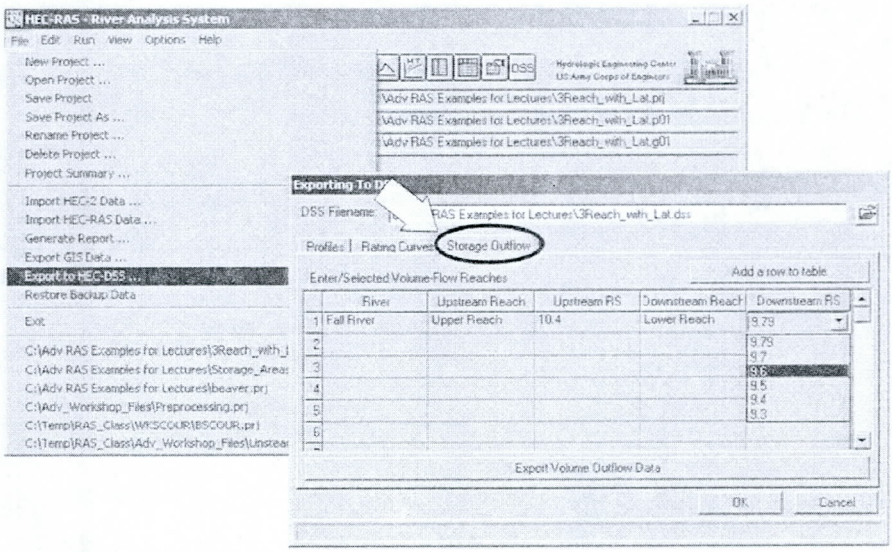
WEST CONSULTANTS, INC
www.westconsultants.com



Rating curves (stage versus flow) can be exported by selecting the second tab. All that is required to export rating curves is a list of the cross section locations. The left portion of the window allows users to select reaches and then select the river stations of that reach. The button with an arrow would then be used to transfer the highlighted river stations to the selected list box on the right. In the above window, 3 cross sections and 1 lateral structure are in the selected list. When the rating curves for bridges and culverts are exported, a family of rating curves is exported to the DSS file. The family of rating curves is a set of *upstream* stage verses flow curves for a variety of tailwater elevations (HEC). Results from interpolated cross sections can also be exported (April 2001 course notes). The default is to export stage vs. flow in the order in which they occurred in the simulation. The "Sort flows in rating curves" box can be checked to write to DSS results sorted in order of increasing flow (WEST 06.07.02).

Exporting HEC-RAS Results to DSS – Storage Outflow Info

WEST CONSULTANTS, INC
www.westconsultants.com

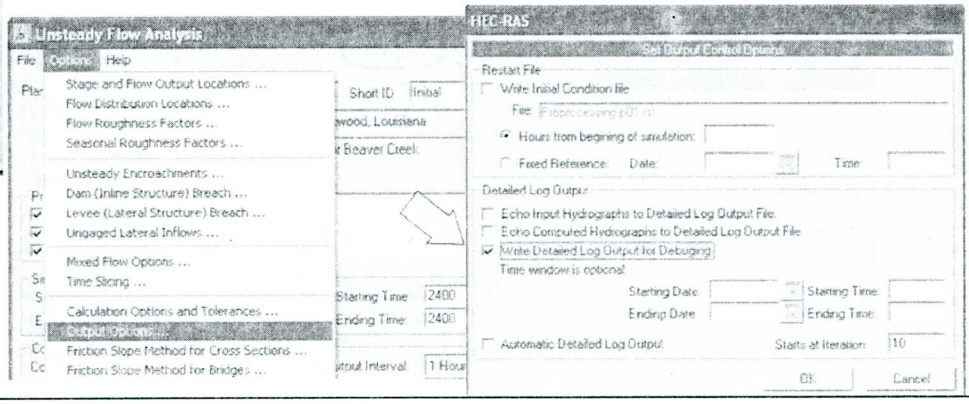


Storage outflow information consists of a table of the computed volume of water between a specified range of cross sections for a given flow. This information can be used for hydrologic routing in a hydrology model such as HEC-HMS or HEC-1. The range of cross sections can span reaches but must start and end on the same river. The starting and ending river stations are entered into the cells in a row of the table. The cells of the table will change to a drop-down list box with the available options listed. For example, if a cell in the first column is selected, the list box will have all the rivers in the system, but if cell in the second column is selected, only the reaches in the river entered in the first column will be present in the drop down list. The DSS pathname made to store the data is based on the downstream cross section location. The A part of the pathname is the combination of the river and reach names, the B part is the downstream river station, and the F part is the plan short ID. For the example above the following DSS pathname would be added: /FALL RIVER UPSTREAM REACH/9.6/FLOW-VOLUME///100 YR EVENT/ (HEC).

Log File Output

WEST CONSULTANTS, INC.
www.westconsultants.com

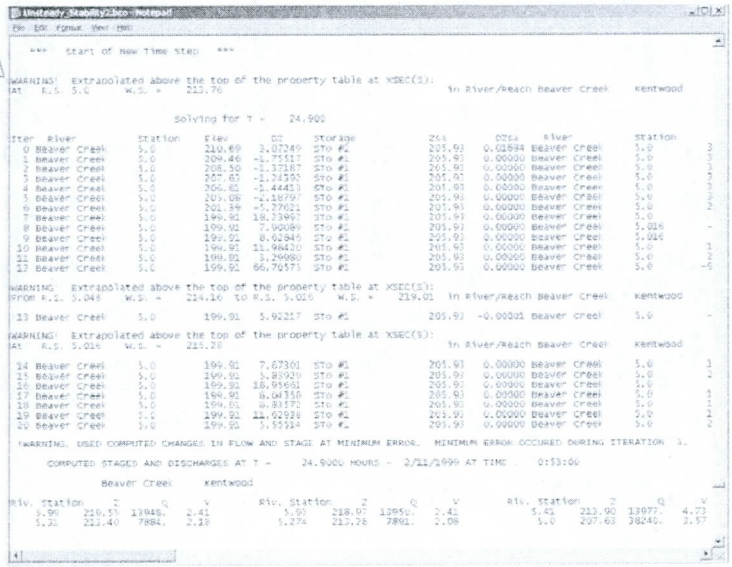
- Contains information about progression of the simulation
- Can be used to help diagnose stability problems
- Viewed in RAS through a text editor



The unsteady flow computation engine can write detailed output to a text log file. The log file contains a detailed information about the progression of computations and can be of assistance in finding errors that cause the unsteady flow program to crash. The log file can easily get into the hundreds of megabytes in size. If your simulation is crashing a long way into it, consider creating a restart file before things go unstable. Then restart the simulation nearer the point where it crashes. This output option is turned on from the **Options** menu on the Unsteady Flow Analysis Editor. Select **Output Options** and a dialog will appear (right) with an option to turn on the detailed output. When this option is on, the unsteady flow computation engine will write a text file that can be accessed from the options menu by selecting **View Computation Log File** (HEC).

Example Log File Output

WEST CONSULTANTS, INC.
www.westconsultants.com



```
*** Start of New Time Step ***

WARNING! Extrapolated above the top of the property table at XSEC(S):
At  R.S. 5.0     W.S. =    215.76                    in River/Reach Beaver Creek     Kentwood

                    Solving for T =    24.900

Iter  River          Station    Elev        DZ      Storage          Zss        DZss    River        Station
  0 Beaver Creek     5.0        210.69   3.07249    Sto #1           205.93   0.01694 Beaver Creek   5.0        3
  1 Beaver Creek     5.0        209.46  -1.75517    Sto #1           205.93   0.00000 Beaver Creek   5.0        3
  2 Beaver Creek     5.0        208.50  -1.37187    Sto #1           205.93   0.00000 Beaver Creek   5.0        3
  3 Beaver Creek     5.0        207.63  -1.24593    Sto #1           205.93   0.00000 Beaver Creek   5.0        3
  4 Beaver Creek     5.0        206.81  -1.44413    Sto #1           205.93   0.00000 Beaver Creek   5.0        3
  5 Beaver Creek     5.0        205.08  -1.18279    Sto #1           205.93   0.00000 Beaver Creek   5.0        2
  6 Beaver Creek     5.0        201.54  -5.27021    Sto #1           205.93   0.00000 Beaver Creek   5.0        2
  7 Beaver Creek     5.0        199.91  18.23997    Sto #1           205.93   0.00000 Beaver Creek   5.0
  8 Beaver Creek     5.0        199.91   7.90089    Sto #1           205.93   0.00000 Beaver Creek   5.016      -
  9 Beaver Creek     5.0        199.91   8.02846    Sto #1           205.93   0.00000 Beaver Creek   5.016
 10 Beaver Creek     5.0        199.91  11.98420    Sto #1           205.93   0.00000 Beaver Creek   5.0        1
 11 Beaver Creek     5.0        199.91   3.26980    Sto #1           205.93   0.00000 Beaver Creek   5.0        2
 12 Beaver Creek     5.0        199.91  66.70171    Sto #1           205.93   0.00000 Beaver Creek   5.0        -1

WARNING  Extrapolated above the top of the property table at XSEC(S):
From R.S. 5.048    W.S. =    224.16  to R.S. 5.016    W.S. =    219.01  in River/Reach Beaver Creek     Kentwood

 13 Beaver Creek     5.0        199.91   5.92217    Sto #1           205.93  -0.00001 Beaver Creek   5.0        -

WARNING! Extrapolated above the top of the property table at XSEC(S):
At  R.S. 5.016     W.S. =    215.28                    in River/Reach Beaver Creek     Kentwood

 14 Beaver Creek     5.0        199.91   7.67301    Sto #1           205.93   0.00000 Beaver Creek   5.0        1
 15 Beaver Creek     5.0        199.91   5.83929    Sto #1           205.93   0.00000 Beaver Creek   5.0        -
 16 Beaver Creek     5.0        199.91  18.95661    Sto #1           205.93   0.00000 Beaver Creek   5.0
 17 Beaver Creek     5.0        199.91   8.04358    Sto #1           205.93   0.00000 Beaver Creek   5.0        1
 18 Beaver Creek     5.0        199.91   6.83572    Sto #1           205.93   0.00000 Beaver Creek   5.0        1
 19 Beaver Creek     5.0        199.91  11.92928    Sto #1           205.93   0.00000 Beaver Creek   5.0        1
 20 Beaver Creek     5.0        199.91   5.55214    Sto #1           205.93   0.00000 Beaver Creek   5.0        2

 WARNING. USED COMPUTED CHANGES IN FLOW AND STAGE AT MINIMUM ERROR.  MINIMUM ERROR OCCURED DURING ITERATION  3.

     COMPUTED STAGES AND DISCHARGES AT T =     24.9000 HOURS -  2/21/1999 AT TIME     0:53:00

          Beaver Creek     Kentwood

Riv. Station     Z        Q        V        Riv. Station     Z        Q        V        Riv. Station     Z        Q        V
   5.99    219.55  13946.   2.41             5.95    218.97  13950.   2.41             5.41    212.90  13977.   4.73
   5.32    212.40   7884.   2.18             5.274   213.28   7891.   2.08             5.0     207.63  38240.   3.97
```

25

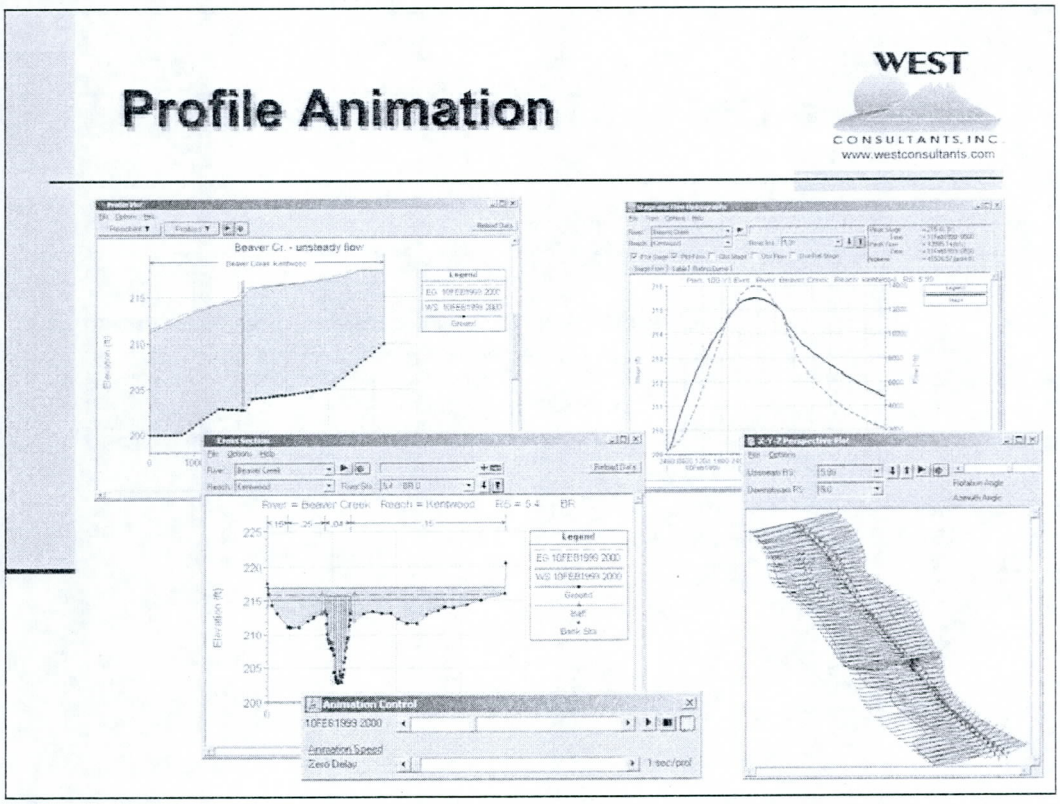
This is the *.BCO file from UNET. SEC ID (e.g., 1s6.000) are identified by reach number (e.g., 1), followed by an 's' and then the RS (e.g., 6.000). This is a good place to look for the word "warning". This is discussed in more detail in the debugging and stability lecture (WEST 05.02.02).

Post-Processor

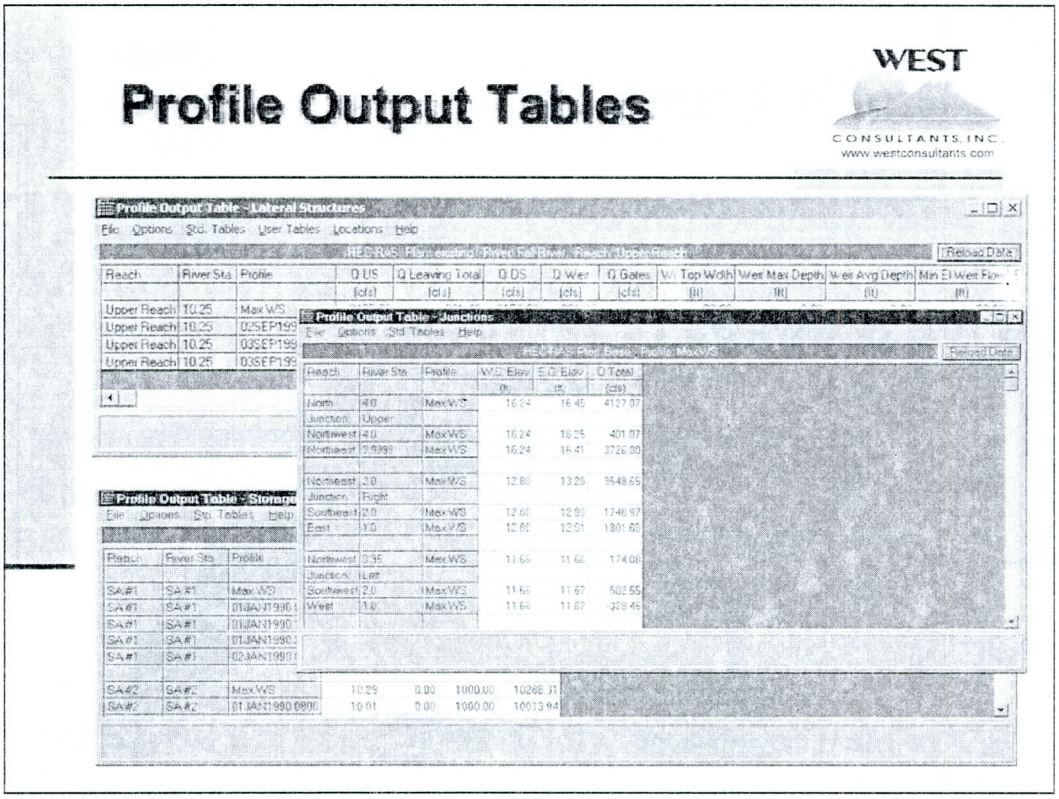
WEST CONSULTANTS, INC.
www.westconsultants.com

- Can be run after the unsteady simulation is completed
- Provides profiles for the maximum stage and at regular intervals
- All regular graphics and tables can be used to view the post process results
- Generates data for animation

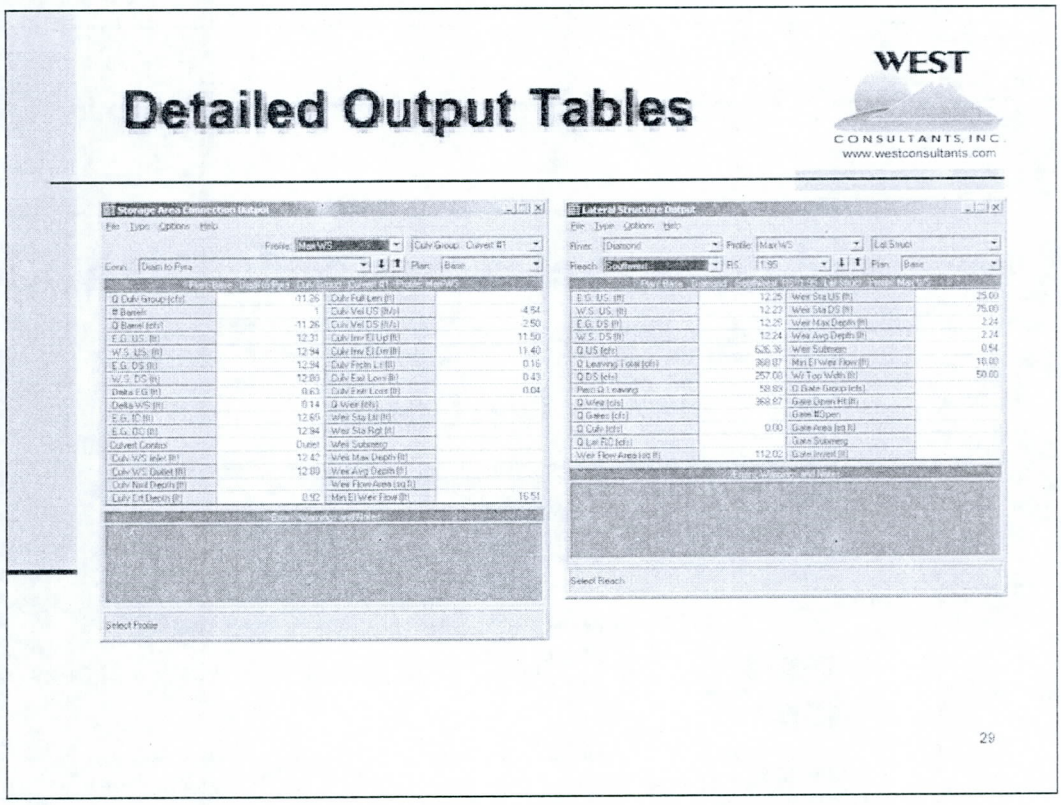
26



The cross section plots, profiles, and 3-D plots can be animated. Under the **Options** menu of the profile plot, select **Animate** and the animation bar will be displayed. Select the button with an arrow to the right for an animation. The animation will affect all graphics loaded, so if a cross section and profile plot are opened, the water surface will be animated in both windows (HEC).



Profile summary tables are used to show a limited number of hydraulic variables for several cross sections (HEC). The "Max WS" is the maximum water surface from the simulation period and can be the data from various time steps at different cross sections (WEST 09.23.03).



Detailed output tables show hydraulic information at a single location, for a single profile (HEC).

WEST

CONSULTANTS, INC.
www.westconsultants.com

Additional Geometric Data for RAS Unsteady Flow Models

- **Objective**: This lecture will discuss supplementary unsteady flow geometric data, in addition to what is typically specified in a RAS steady flow model.

This lecture was originally developed by the staff and engineers at the US Army Corps of Engineers Hydrologic Engineering Center (HEC).

1

Topics to Discuss

WEST CONSULTANTS, INC. www.westconsultants.com

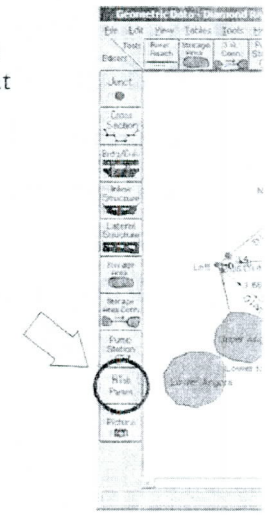
- Additional data requirements and options for RAS unsteady flow modeling include:
  - Generating hydraulic properties tables (HTab)
  - Permanent vs. non-permanent ineffective flow areas
  - Pilot channels

2

Hydraulic Tables and the Geometric Pre-Processor

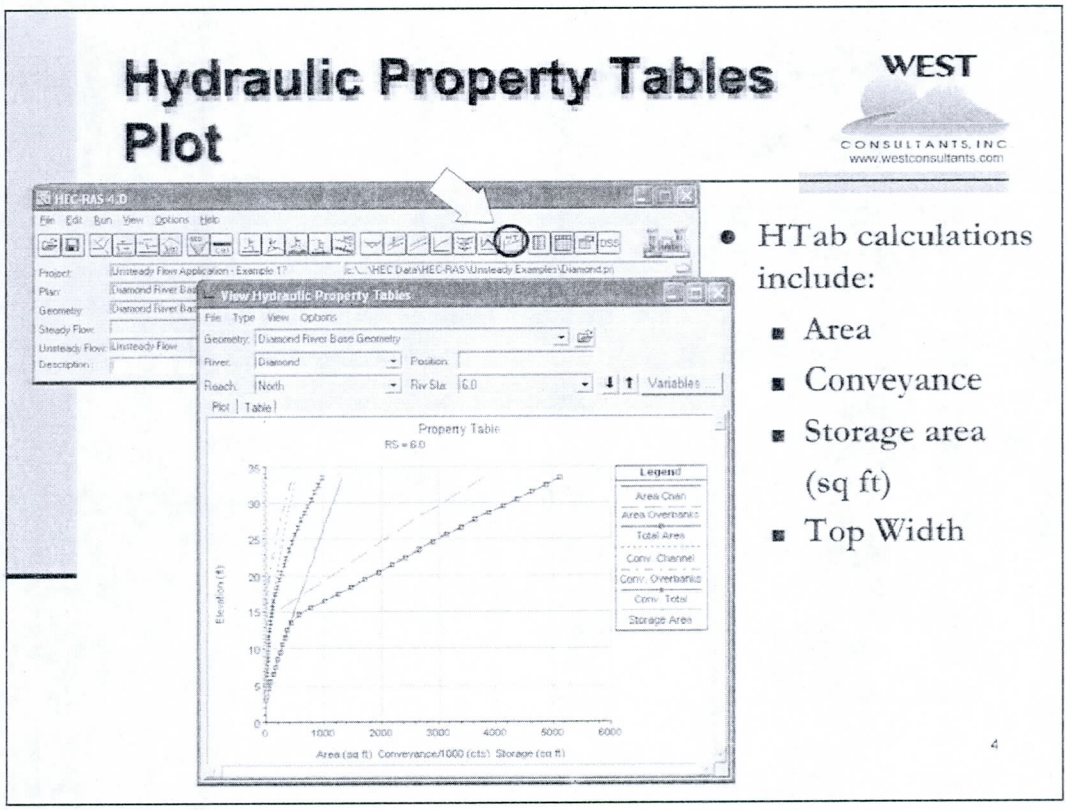
WEST
CONSULTANTS, INC
www.westconsultants.com

- What does it do?
  - Processes geometric data into a series of hydraulic tables and rating curves at cross sections and structures
  - Computations made prior to, and independent of, unsteady flow calculations
- Why use it?
  - RAS interpolates hydraulic variables from the tables during the unsteady flow calculations
  - Saves computation time - faster than computing the values during each iteration



3

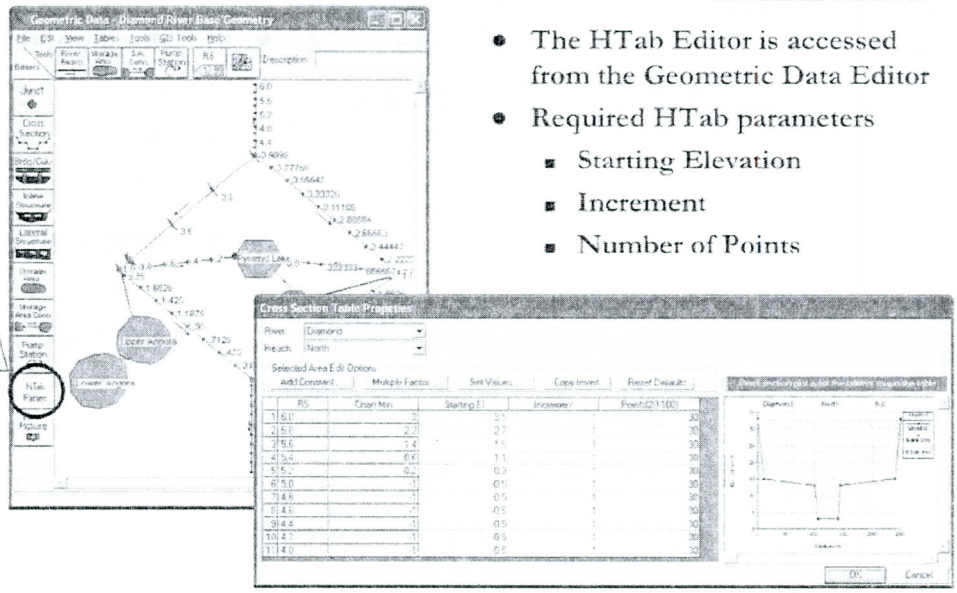
The geometric pre-processor is used to process the geometric data into a series of hydraulic property tables and rating curves. This is done in order to speed up the unsteady flow calculations. Instead of calculating hydraulic variables for each cross section during each iteration, the program interpolates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



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. Viewing the graphical output is a good diagnostic tool for examining cross section geometry. The output is located under the **View** menu, or the shaded button in the figure. 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. Clicking on the "Table" tab will show the tailwater information in tabular form. 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 (HEC). A "Warning! Extrapolated above the top of the property table" will appear in the Computation Log file if the water surface elevation is computed above the highest elevation of the hydraulic table (WEST 06.07.02).

Entering Cross Section HTab Values

WEST CONSULTANTS, INC
www.westconsultants.com



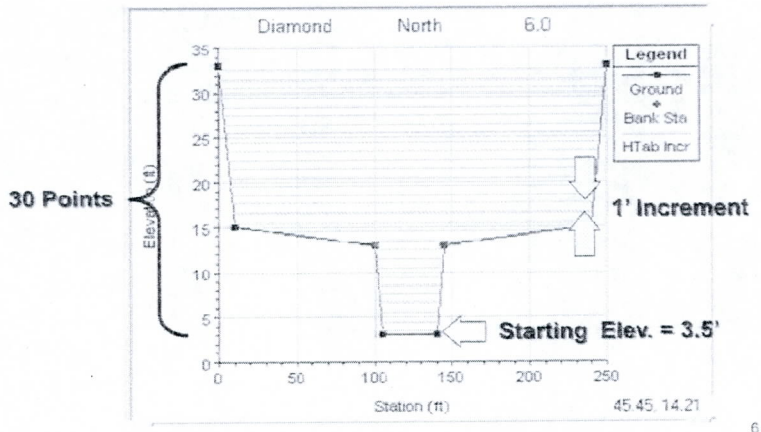
- The HTab Editor is accessed from the Geometric Data Editor
- Required HTab parameters
  - Starting Elevation
  - Increment
  - Number of Points

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. 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 (HEC). **Default Parameters** (English): Starting elevation = 0.5' above minimum cross-section ground point; Increment = a function of the vertical (elevation) range of geometry points, up to 1' maximum, Number of Points = number of points required to span the vertical (elevation) range of geometry points for the determined increment, a minimum of 20 and a maximum of 100. To revert back to default values, delete data from all columns (WEST 05.17.02)

Setting HTAB Parameters

WEST CONSULTANTS, INC.
www.westconsultants.com

	RS	Chan Min	Starting El	Increment	Points[20-100]
1	6.0	3	3.5	1	30

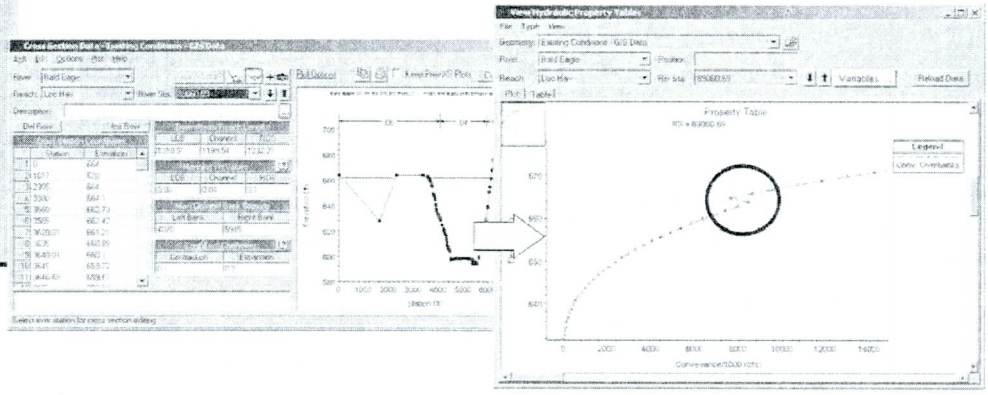


6

Increment values should be set small enough so that the HTAB values adequately reflect the channel geometry.

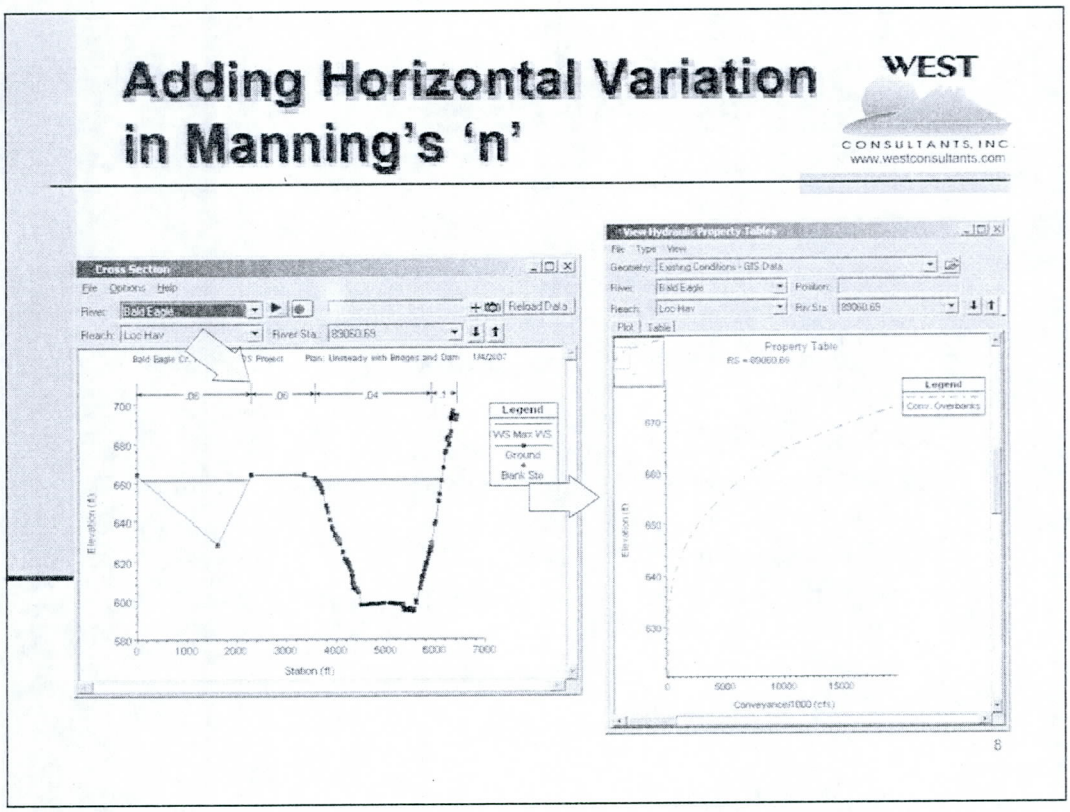
Smooth Transitions in HTab values

WEST
CONSULTANTS, INC
www.westconsultants.com



7

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, 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 just below the flat terrace on the left overbank (HEC). Smooth transitions are desired in all of the hydraulic property tables. Abrupt changes can occur when the overbank ground slope becomes relatively flat, causing a sudden change in conveyance. A work around is to set a horizontal change in 'n' value at this grade break. Note, however, that the Manning's 'n' value does not need to change. Separate conveyance calculations are made for each overbank, the channel, as well as for any sections with changes in Manning's 'n' (this is discussed in Chap. 2 of the Hydraulic Reference Manual) (WEST 05.17.02).



To account for this situation, a subdivision made by inserting another Manning's 'n' value could 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 (HEC). The sudden change in conveyance is removed by adding the horizontal change in Manning's 'n'. (WEST 05.17.02).

Ineffective Flow Areas

WEST CONSULTANTS, INC www.westconsultants.com

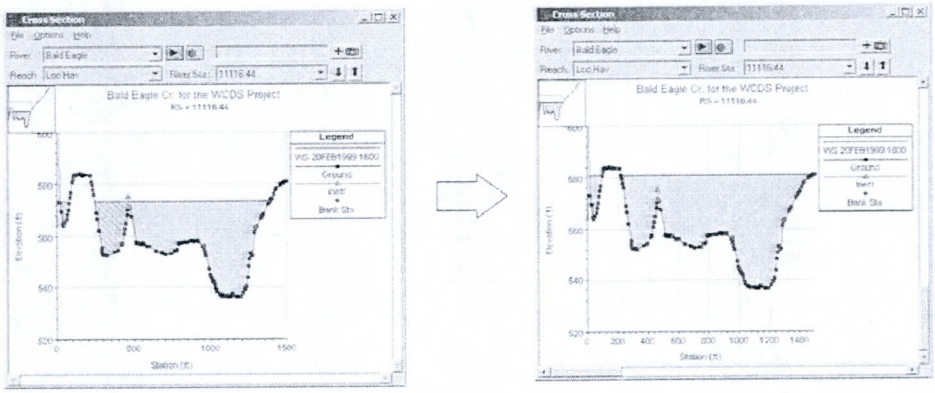
- For ineffective flow areas:
  - Water is not actively being conveyed
  - Velocity is close to zero in the downstream direction
- Unsteady flow models account for:
  - Storage
  - Conveyance
- When the water surface is below the (non-permanent) ineffective flow area elevation:
  - Ineffective area = storage
- When the water surface is above the (non-permanent) ineffective flow area elevation:
  - Ineffective area = active conveyance

9

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, storage is not simulated, and portions of the cross section are typically blocked off with ineffective areas. It may be useful to think of storage-only portions of a cross section as having extremely high n-values, which effectively nullifies the conveyance. 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 (HEC).

Non-Permanent Ineffective Flow Areas

WEST CONSULTANTS, INC
www.westconsultants.com

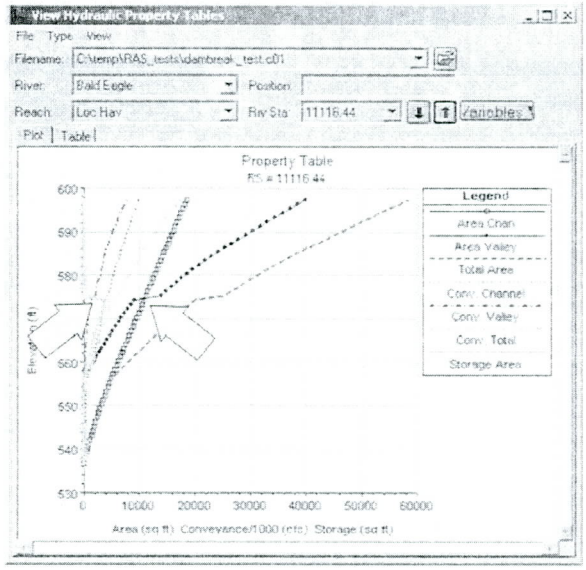


10

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 (HEC).

Effects of Overtopping Non-Permanent Ineffective Areas

WEST CONSULTANTS, INC
www.westconsultants.com

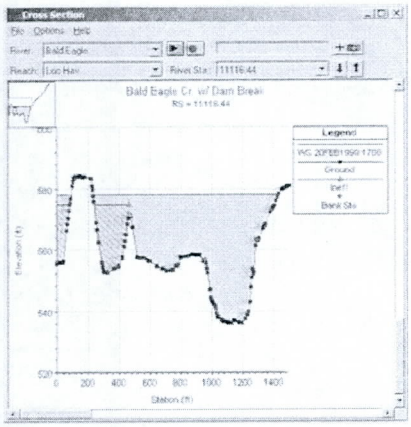


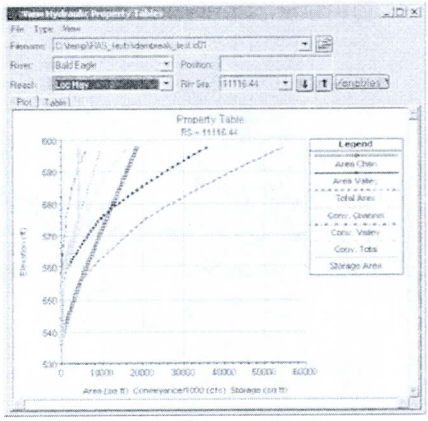
11

Storage goes to zero while conveyance increases dramatically, when non-permanent ineffective areas are overtopped (WEST 05.17.02).

Permanent Ineffective Flow Areas

WEST CONSULTANTS, INC
www.westconsultants.com



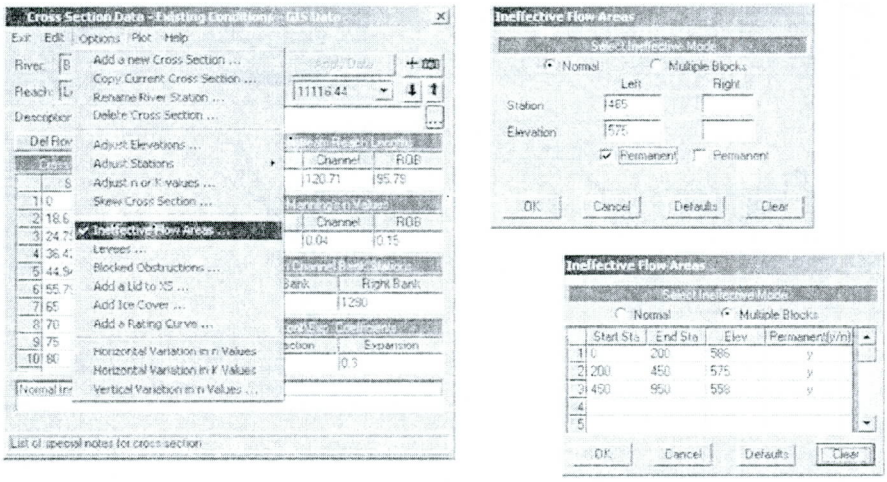


12

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 (HEC).

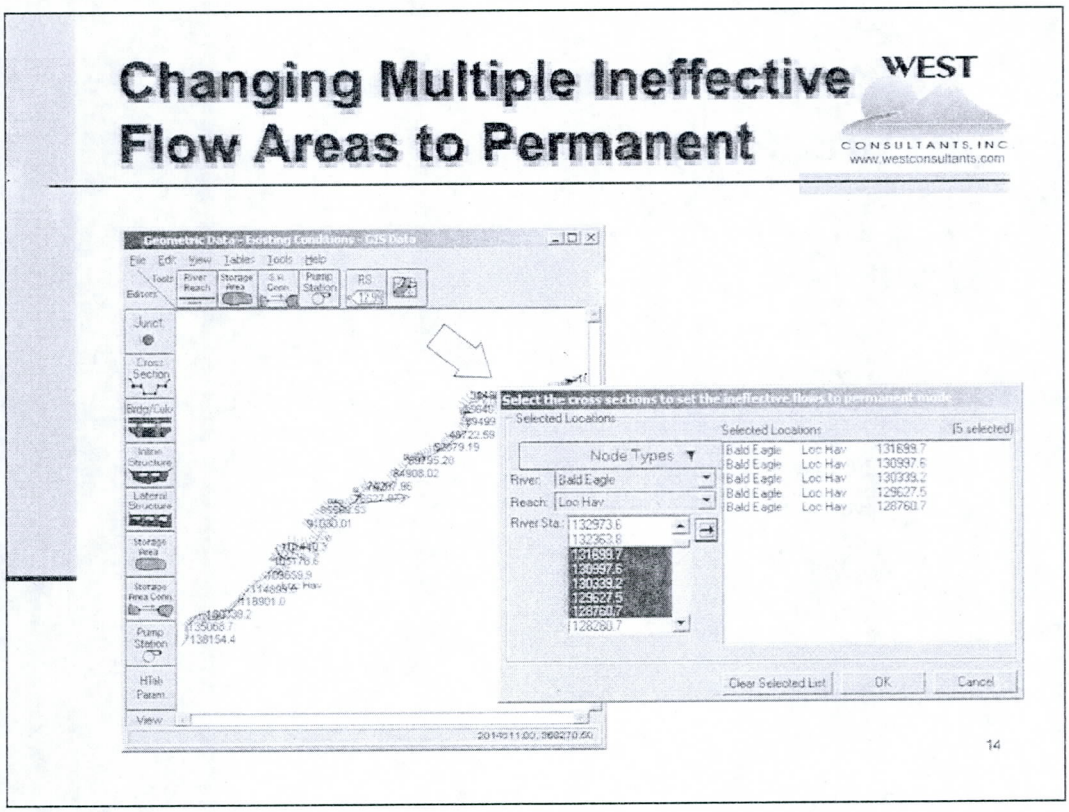
Setting Permanent Ineffective Flow Areas

WEST CONSULTANTS, INC.
www.westconsultants.com



13

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

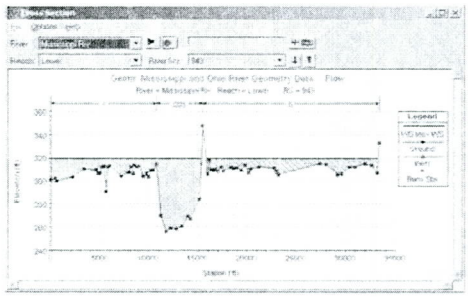


The default in HEC-RAS is to set ineffective flow areas to non-permanent. To change a set of river stations from non-permanent to permanent, select **Set Ineffective Areas to Permanent Mode** under the **Tools** menu located in the Geometric Data Editor. Then select the proper river, reach, and river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

Modeling Overbank Areas in Unsteady Flow

WEST CONSULTANTS, INC
www.westconsultants.com

- At what elevation is the right bank:
  - Storing water?
  - Conveying flow?
- Modeling options:
  - Non-permanent ineffective flow area
  - Permanent ineffective flow area
  - Levee
  - Storage area

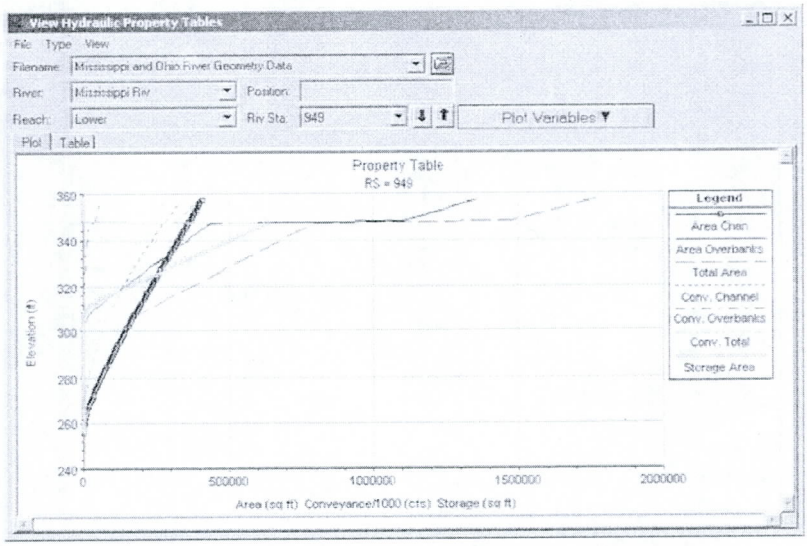


15

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. 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 (HEC).

Non-Permanent Ineffective Flow Areas

WEST CONSULTANTS, INC.
www.westconsultants.com

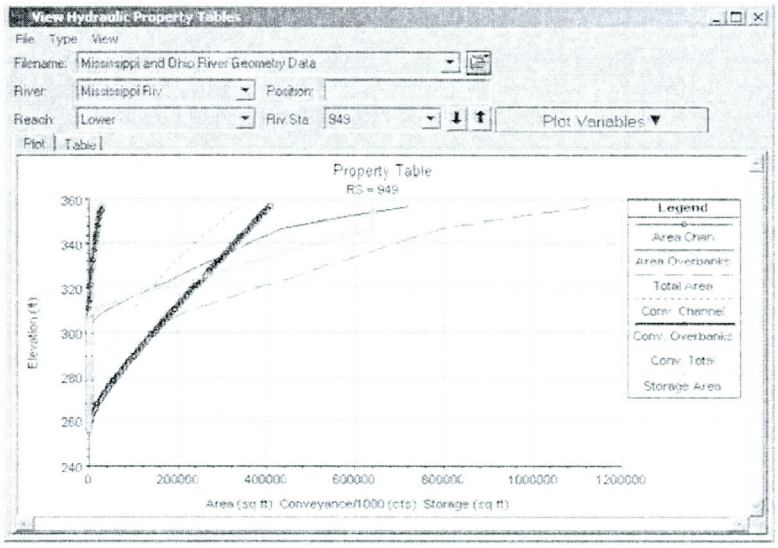


16

The figure above represents the previous cross section with the right overbank modeled as non-permanent ineffective flow. As seen in 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 (HEC).

Permanent Ineffective Flow Areas

WEST CONSULTANTS, INC
www.westconsultants.com



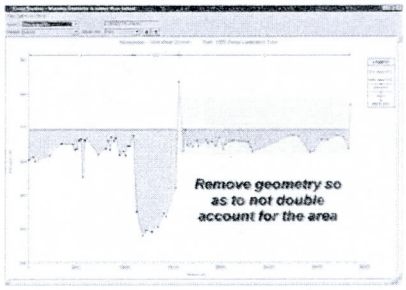
17

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. 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. 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 (HEC). If the levee is overtopped then the user should consider adding a storage area (discussed in a later lecture). Using a combination of levee with a permanent ineffective area (i.e., representing the storage available when the levee is overtopped) might create model instabilities by suddenly creating a large volume available for storage. Also note, if using a levee with a permanent ineffective area, the storage volume would be a function of the right bank geometry defined at this cross section and the distance to the upstream and downstream cross section (WEST 05.17.02).

Storage Area Option

WEST CONSULTANTS, INC
www.westconsultants.com

- Creating a storage area may be appropriate when storage volume is important
  - Terminate the cross section at the beginning of the storage area
  - Add a lateral structure and storage area

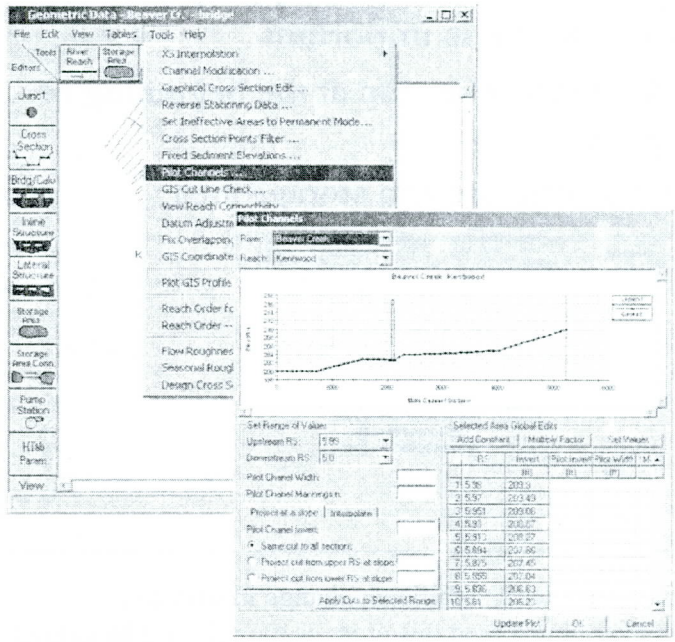


18

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 (HEC). The right overbank should be removed so that the storage in the area is not double counted. A blocked obstruction set to elevation above the maximum water surface elevation should not be used to remove the area since this will add wetted perimeter once the bank (levee) elevation is exceeded (WEST 05.17.02).

Low Flow Pilot Channels

WEST CONSULTANTS, INC.
www.westconsultants.com



- Pilot channels should be used to:
  - Ensure that the reach remains wet
  - Improve stability for low flow

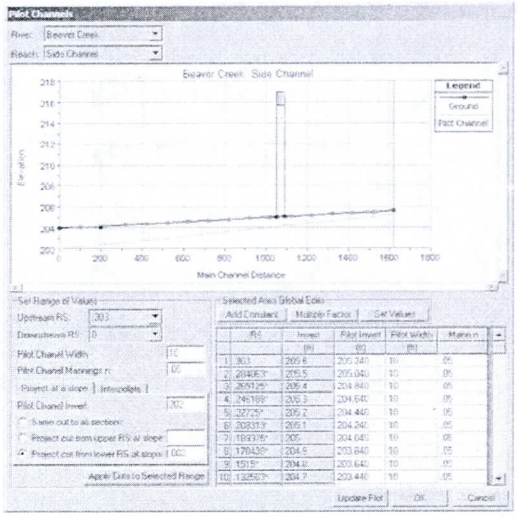
19

A pilot channel cuts a rectangular notch into the bottom of the cross section. Generally this notch is not very wide (often 1 foot is used), but it provides depth to the cross section at low flows (typically make it 5 to 10 feet deep). Additionally, the use of a pilot channel can smooth-out irregularities in the channel bottom. This also helps the stability of the model solution (particularly at very low flow). The pilot channel area and conveyance are "borrowed" from the lower portion of the main channel, such that the total area and conveyance properties of the cross section relate to the original cross section at higher flows. Once the depth of the flow gets higher, the area and conveyance of the pilot channel are ignored (RAS User's Manual, Ver. 3, Jan 2001, Chapter 6). Pilot channels can be added from the **Tools** drop down menu in the Geometric Data Editor. See the UNET Manual for how the pilot channel area is "borrowed" from the main channel (WEST 05.17.02).

Adding a Pilot Channel

WEST CONSULTANTS, INC.
www.westconsultants.com

- Specify
  - Range
  - Width
  - Manning's 'n'
  - Slope or elevations and interpolate
- Individual cross sections can be modified



20

The user selects a river, reach, and range of river stations to apply the pilot channel to. On the lower left hand side of the Pilot Channels Editor are utilities to enter the pilot channel information. The user enters the pilot channel width and the Manning's 'n' value (should be equal to or higher than the main channel 'n' value). The user can either enter an elevation for the invert of the pilot channel and project it on a slope over the range of cross sections, or they can enter an upstream and a downstream invert elevation and have the program use linear interpolation for the cross sections in between. A list of the final pilot channel values for each of the cross sections is shown in the table on the lower right hand side of the editor. The user can modify the table directly and change any value on a cross section by cross section basis. The profile plot on the editor will display the invert elevation of the pilot channel so you can compare it to the actual channel invert. Once you have finished adding the pilot channel information, press the OK button, and then save the geometric data (RAS User's Manual, Ver. 3, Jan 2001, Chapter 6).

WEST

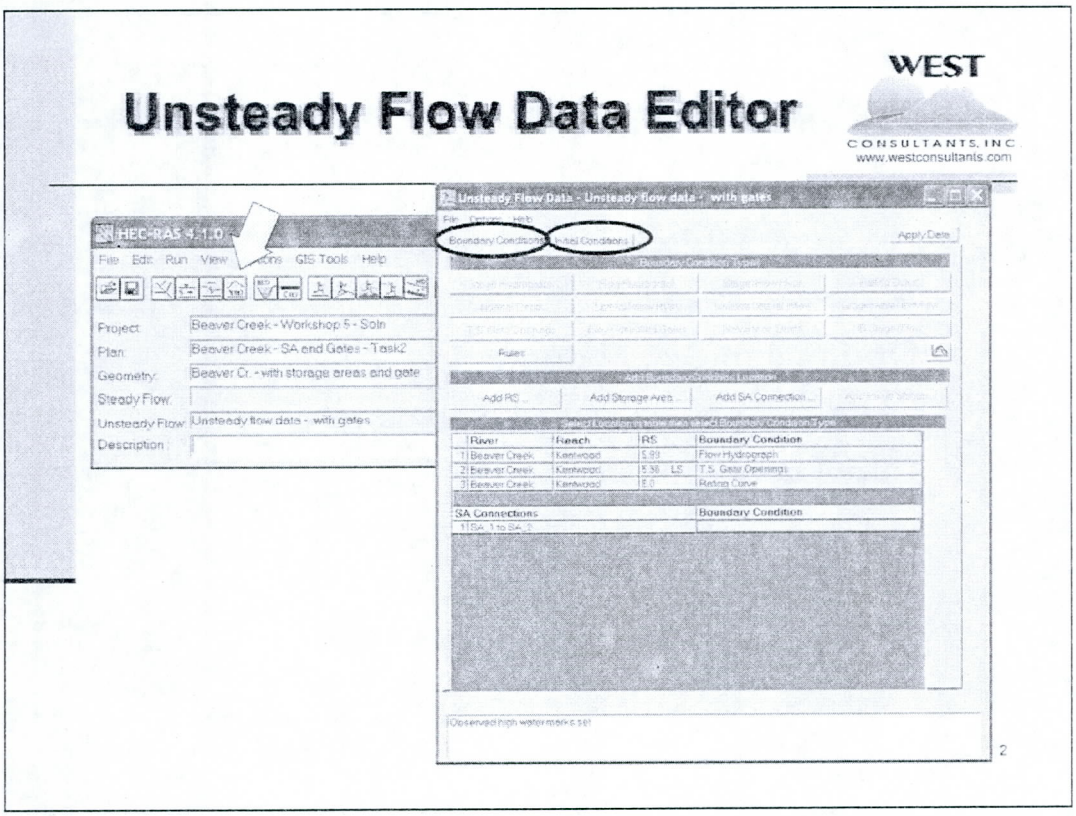
CONSULTANTS, INC.
www.westconsultants.com

Boundary and Initial Conditions in the RAS Unsteady Flow Data Editor

- **Objective**: This lecture will discuss the various unsteady flow boundary and initial conditions and show you how to utilize them in RAS.

This lecture was originally developed by the staff and engineers at the US Army Corps of Engineers Hydrologic Engineering Center (HEC).

References: *HEC-RAS River Analysis System, User's Manual*, Chapter 8, Performing an Unsteady Flow Analysis, January 2001 (HEC).



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. 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 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 (HEC).

Topics to Discuss

WEST CONSULTANTS, INC. www.westconsultants.com

- External boundaries required
  - Upstream and downstream extents of study area
- Optional internal boundaries
  - Inflows not at external boundaries
- Initial conditions required
  - Used for computations at the start of the simulation
- Optionally adding observed data

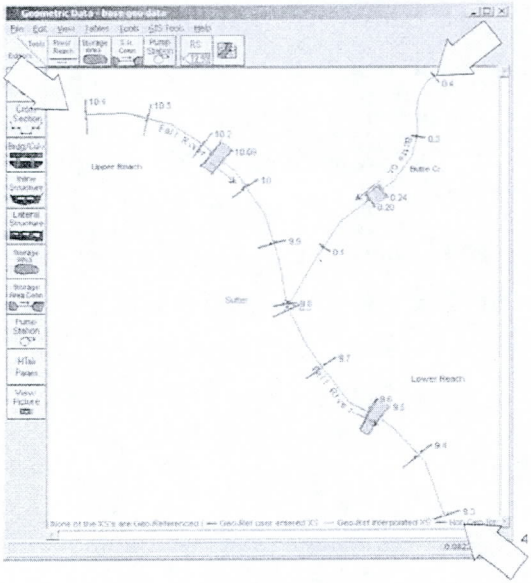
3

The Unsteady Flow Editor defines: external boundary type and data, optional internal boundary data, and initial conditions for the start of the simulation (WEST 05.10.02).

External Boundary Conditions

WEST
CONSULTANTS, INC.
www.westconsultants.com

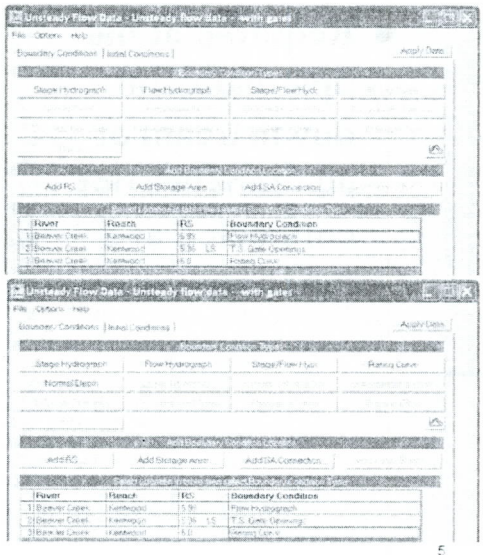
- Required at the upstream and downstream of all branches not connected at a junction



External Boundary Options

WEST CONSULTANTS, INC
www.westconsultants.com

- Upstream options
  - Stage hydrograph
  - Flow hydrograph
  - Stage/flow hydrograph
- Downstream options
  - Stage hydrograph
  - Flow hydrograph
  - Stage/flow hydrograph
  - Rating curve
  - Normal depth



5

The upstream 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 and Flow hydrograph - e.g., combined observed stage and forecasted flow

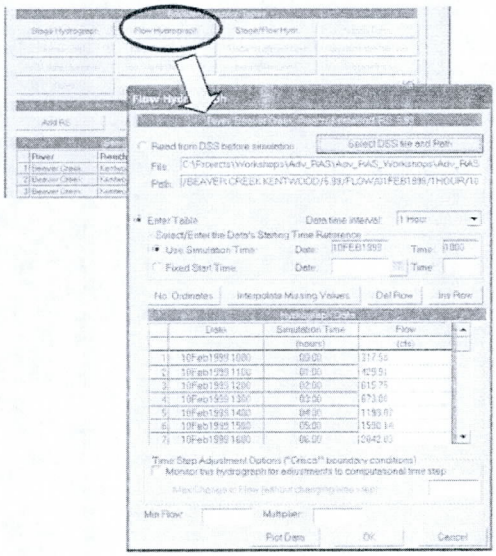
The downstream boundary condition can also be:

- 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

The program will automatically gray-out the boundary condition types that are not relevant (HEC).

External Boundary Condition – Flow Hydrograph

WEST CONSULTANTS, INC
www.westconsultants.com



- Read from DSS
  - Select DSS file
  - Select pathname
- Manual table entry
  - Select time interval
  - Select start date/time
  - Type flow data or copy and paste
  - Interpolate option

6

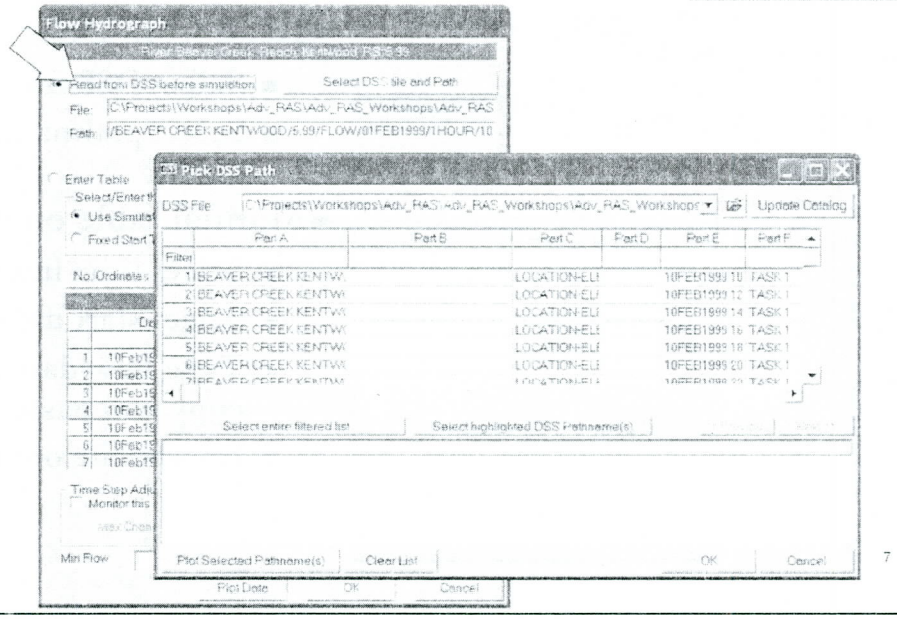
A flow hydrograph can be used as either an upstream boundary or downstream boundary condition, but is most commonly used as an upstream boundary condition. When the flow hydrograph button is pressed, the window shown above will appear. As shown, the user can either read the data from a HEC-DSS (HEC Data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

Stage hydrographs are most commonly used downstream boundaries. Data for stage hydrographs are entered in an identical way to that for flow hydrographs (WEST 02.10.02).

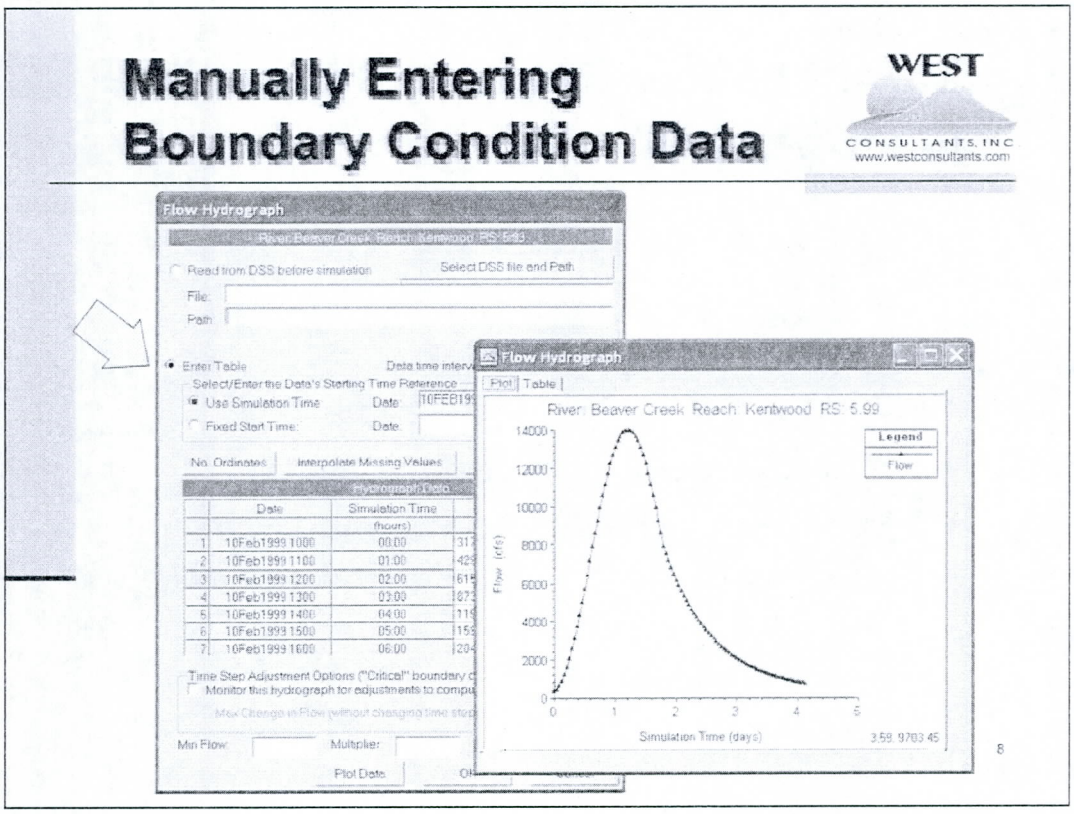
Two other options at the bottom of this editor are "Min Flow" and "Multiplier". Both of these options apply to user entered hydrographs or hydrographs read from HEC-DSS. The "Min Flow" option allows the user to specify a minimum flow to be used in the hydrograph. This option is very useful when too low of a flow is causing instability problems. Rather than edit the user entered hydrograph or the DSS file (depending upon where the hydrograph is coming from), the user can enter a single value, and all values below this magnitude will be changed to that value. The "Multiplier" option allows the user to multiply every ordinate of the hydrograph by a user specified factor. This factor will be applied to the user-entered hydrograph or a hydrograph read from HEC-DSS. (HEC).

Selecting DSS Data for a Boundary Condition

WEST CONSULTANTS, INC.
www.westconsultants.com



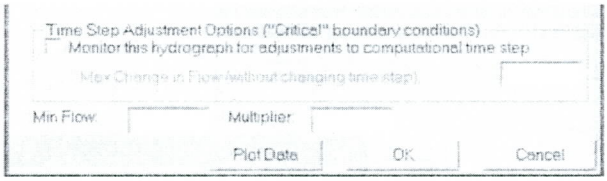
If the user selects the option to read the data from DSS, they must press the "Select DSS File and Path" button. When this button is pressed, a DSS file and pathname selection screen will appear as shown. The user first selects the desired DSS file by using the browser button at the top. Once a DSS file 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 (HEC).



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 are 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 are selected, the user can then begin entering the data. The "Simulation Time" column will always start with 00:00 hours. For example, if 00:15 was entered as the fixed start time, the "Date" column would read 09Feb1999 0015, but the "Simulation Time" column would read 00:00. Fixed start times should be entered in military time format, e.g., "0015" to start at 15 minutes into a new day. Entering just "15" will result in a start time of 1500 hours (HEC). The "Plot Data" button can be used to view either the manually entered data or the specified DSS file (WEST 05.10.02).

Monitor Flow Rate-of-Change

WEST CONSULTANTS, INC
www.westconsultants.com



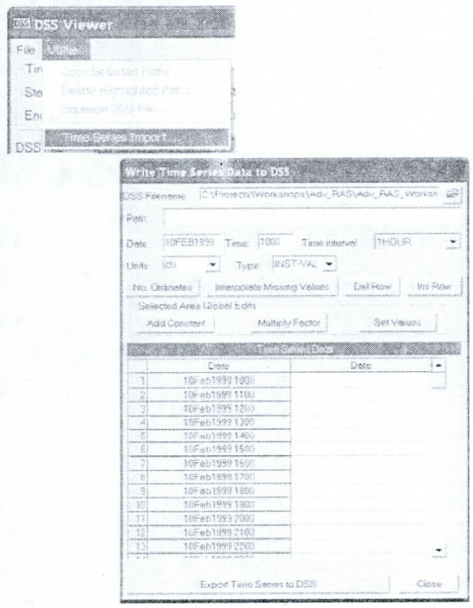
- Added option for flow hydrograph data
- Set maximum flow change in time step
- If input data exceeds the maximum value, the program will shorten computational time step

9

An additional option listed on the flow hydrograph boundary condition is to make this boundary a "Critical Boundary Condition." 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 maximum. Large changes in flow can cause instabilities. The use of this feature can help to keep the solution of the program stable (HEC).

Entering Regular Time Series Data into a DSS file

WEST CONSULTANTS, INC
www.westconsultants.com



- Specify
  - Pathname
  - Start date and time
  - Time interval
  - Units
  - Data type
- Type data or copy and paste

10

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 (HEC). From the main RAS project definition window, click the "DSS button", then "Utilities", and "Time Series Import". Specify the DSS file to write to, the required DSS data (example path name formatting: "/A_PART/B_PART/FLOW/01APR1969/1HOUR/OBS/"), and the data values. Then click "Export Time Series to DSS". A pop-up window will appear to let the User know that the data was successfully written to DSS (WEST 02.10.02).

Sources of Time-Series Data

WEST CONSULTANTS, INC. www.westconsultants.com

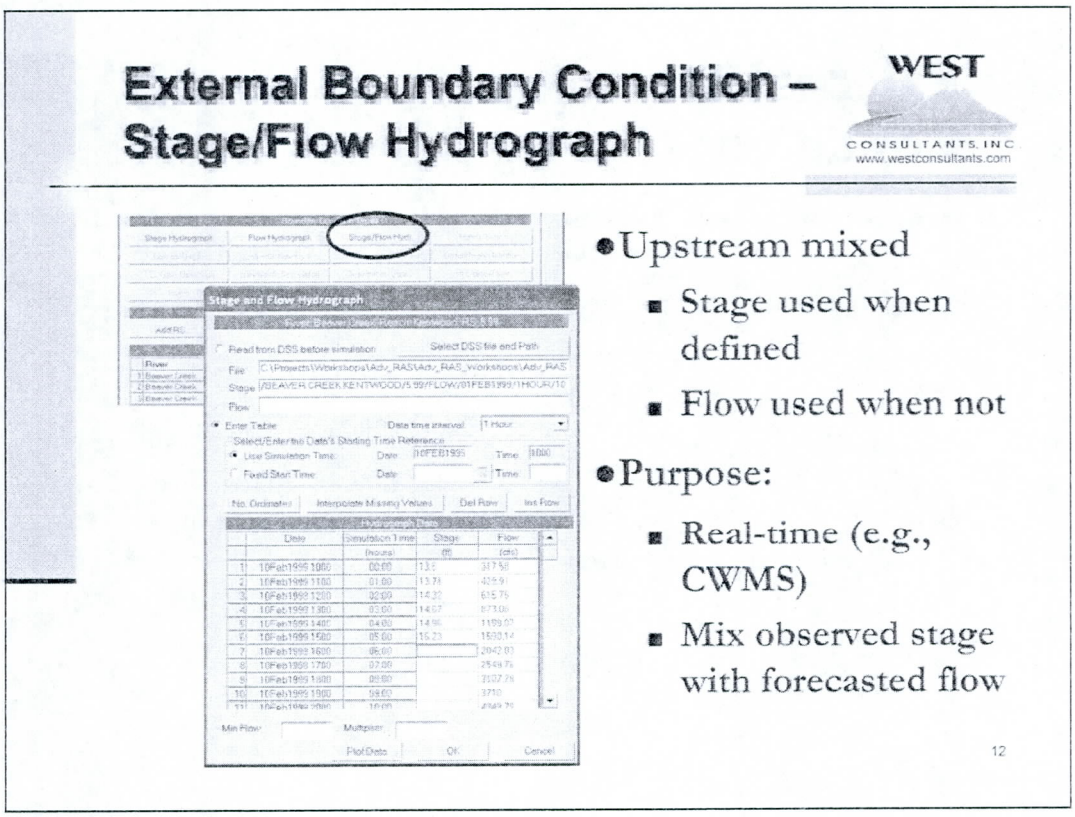
- USGS (http://water.usgs.gov)
  - Stage and flow hydrographs
- Other federal, state, county, or local agencies
- Computed synthetic floods
- Rainfall-runoff models (e.g., HEC-HMS)
- Peak discharge with assumed time distribution

11

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 users, 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 U.S. Geological Survey
- 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).

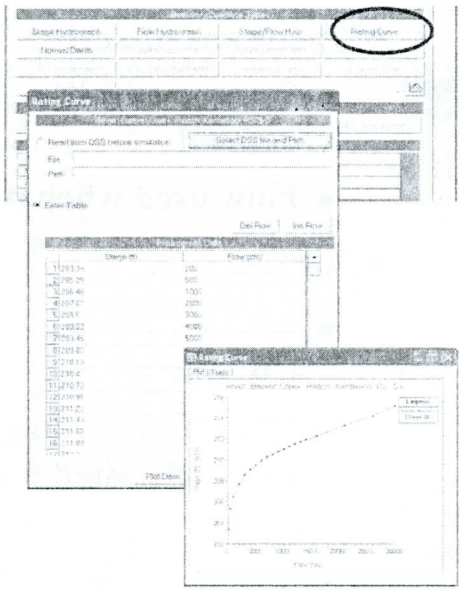
USGS provides data online at intervals as short as 15 minutes for recent storm events. Other specific agencies that may have data for your study area include: U.S. Bureau of Reclamation, Department of Ecology, Water Management and Irrigation Districts (WEST 11.19.03)



Stage and Flow Hydrograph: The stage and flow hydrograph option can be used together as either an upstream or downstream boundary condition. The upstream stage and flow hydrograph is a mixed boundary condition 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 (HEC).

External Boundary Condition – Rating Curve

WEST CONSULTANTS, INC
www.westconsultants.com



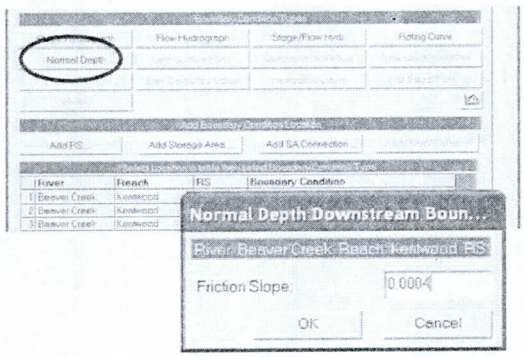
- Single value rating
- Downstream boundary
- Read from DSS
  - Select DSS file
  - Select pathname
- Enter in table
  - Enter stage-flow data or cut and paste

13

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 "looped" part 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 (HEC).

External Boundary Condition – Normal Depth

WEST CONSULTANTS, INC
www.westconsultants.com



- Enter friction (energy) slope
- Program uses Manning's equation to compute stage
- Provides semi-dynamic downstream boundary

14

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 so that any errors it produces will not affect the results at the study reach (HEC). The User might use the local bed slope, or a slope estimated from a USGS quad, if the slope of the friction or water surface are not known (WEST 06.07.02).

Internal Boundary Conditions

WEST CONSULTANTS, INC. www.westconsultants.com

- Types:
  - Lateral inflow hydrograph
  - Uniform lateral inflow
  - Groundwater Interflow
  - Time series gate openings
  - Elevation controlled gates
  - Navigation dams
  - Internal boundary stage and flow hydrograph
  - Rules

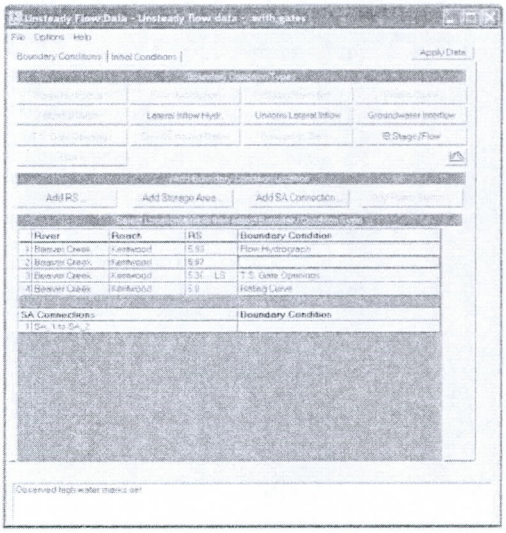
15

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 (HEC).

Adding an Internal Boundary Condition

WEST CONSULTANTS, INC
www.westconsultants.com

- Select:
  - River
  - Reach
  - River station
- Add a boundary condition location

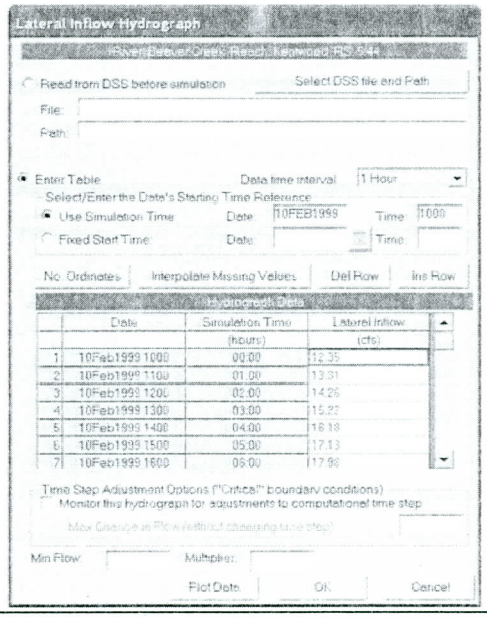


16

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. **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 (HEC).

Internal Boundary Condition – Lateral Inflow Hydrograph

WEST CONSULTANTS, INC
www.westconsultants.com



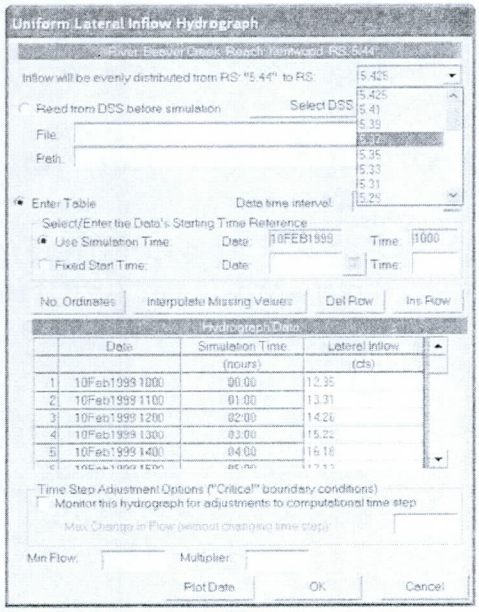
- Flow added at a specific cross section
- Flow data specified in DSS file or entered manually, as for external boundaries

17

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 (HEC).

Internal Boundary Condition – Uniform Lateral Flow Hydrograph

WEST CONSULTANTS, INC. www.westconsultants.com



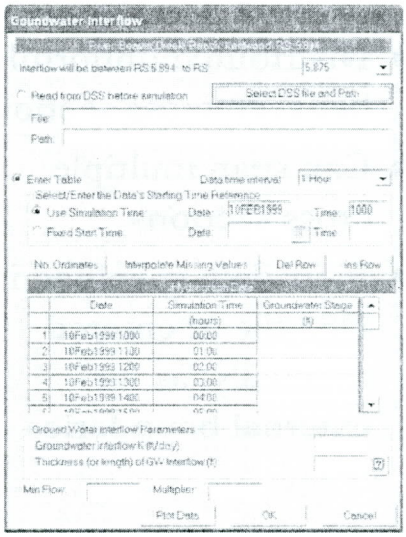
- Distributed uniformly between cross sections
- Can span multiple cross sections
- Flow data specified in DSS file or entered manually, as for external boundaries

18

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). Specifying inflow distributed uniformly between two adjacent XSECs is identical to specifying a lateral inflow (at the upstream of the two XSECs). The distribution of the uniform flow between multiple cross-sections is proportional to the reach length. Uniform lateral flow cannot span multiple river reaches (WEST 06.17.02).

Internal Boundary Condition – Groundwater Interflow

WEST CONSULTANTS INC
www.westconsultants.com



- Distributed uniformly between specified cross sections
- Groundwater parameters required:
  - Darcy's loss coefficient (hydraulic conductivity)
  - Distance over head difference
- Flow data specified in DSS file or entered manually, as for external boundaries

19

Groundwater Interflow: This option allows the user to identify a reach that will exchange water with a specified groundwater reservoir head. The head in 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 exchange is similar to a uniform lateral inflow in that the user enters an upstream and a downstream river station, in which the direction of flow is a function of the head difference. The computed flow is proportional to the head difference between the river and the groundwater reservoir. The computation of the exchange uses Darcy's equation. The user is required to enter Darcy's groundwater loss coefficient (hydraulic conductivity), as well as a time series of heads for the groundwater aquifer. The groundwater aquifer is assumed to be very large such that the interchange of water with the river has no impact on groundwater stage (HEC).

Darcy's Law

WEST CONSULTANTS, INC. www.westconsultants.com

Q = flow (cfs) $\qquad Q = V A$

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

- Top width * length, if groundwater below channel
- Overbank width * length, plus channel depth * length, if groundwater is above channel invert.

V = velocity (ft/hour) $\qquad V = K S$

K = "Darcy's loss coefficient" or hydraulic conductivity (ft/hour)

S = slope (ft/ft) = stage difference / user specified "distance over which head acts"

20

Darcy (1856) applied the principles of fluid flow to the flow of water in a permeable media. Darcy's Law states that 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 Linsley, Kohler, Paulhus)*

- In RAS, the slope is computed as the difference between the water surface elevation at the cross section and the specified groundwater stage (head), divided by the user specified 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.

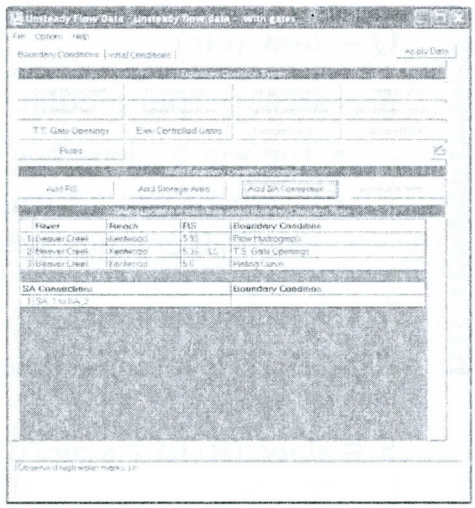
  If the groundwater level is below the invert of the channel, the flow is down from the channel and the width is the average top width.

  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 (HEC).

Internal Boundary Conditions – Gates

WEST
CONSULTANTS, INC
www.westconsultants.com

- Must be specified at a lateral or inline structure
  - "LS" or "IS" after the river station
- Types:
  - Time-series
  - Elevation controlled
  - Navigation dams (IS)
  - Rules



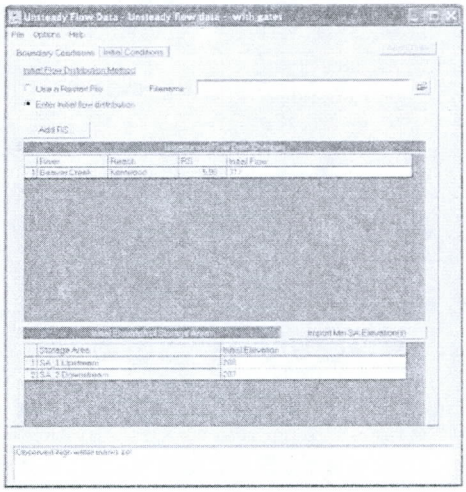
21

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 (HEC). Gates, and setting the internal boundary condition data, are covered in more detail in the inline and lateral weir lecture (WEST 02.10.02). **Navigation Dams.** An internal boundary for optimizing gate control to maintain water surface elevations in reservoirs. **Rules.** This option allows the user to control gate operations based on a set of rules defined by the user. Can control gates by a number of different variables including stages, flows, time, etc.

Initial Conditions

WEST CONSULTANTS, INC
www.westconsultants.com

- Initial flow is required for all reaches
- Initial stage is required for all storage areas
- Flow change locations can be added
- A "restart" file can be read from DSS
  - Specifies stage and flow at each element



22

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. Typically the initial hydrograph stage or flow value is entered when this first option is used. 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 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, RAS uses a dynamic allocation procedure to overcome the time-dimension limit (HEC).

Selecting Initial Conditions Data

WEST CONSULTANTS, INC. www.westconsultants.com

- Initial flow data
  - Should be nearly steady-state conditions
  - For reaches with upstream external flow boundaries, use the initial flow specified in the flow hydrograph
  - Use results from a steady flow simulation
- Initial elevation in storage areas
  - Stage from historic data
- Restart (hot start) file from previous simulation

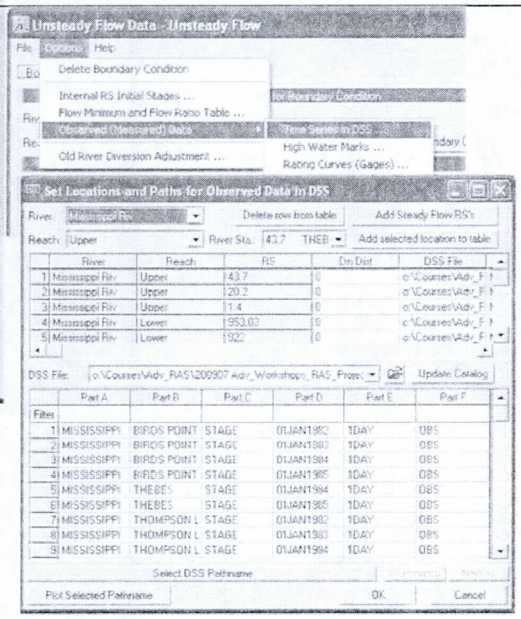
23

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 (HEC).

The **Restart** file can be saved from a simulation and used to resume another simulation. 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 at a reasonable value for the constant flow, those conditions can be written out to a restart file, and then used as the starting conditions (HEC). The User specified initial condition values, and not the data in the boundary condition files, are used to determine the stage and flow in the warm-up period of an unsteady flow simulation. Therefore, it is important that these value match or are at least close to the first boundary condition values in order to maintain model stability (WEST 05.10.02).

Adding Observed Data

WEST CONSULTANTS, INC.
www.westconsultants.com



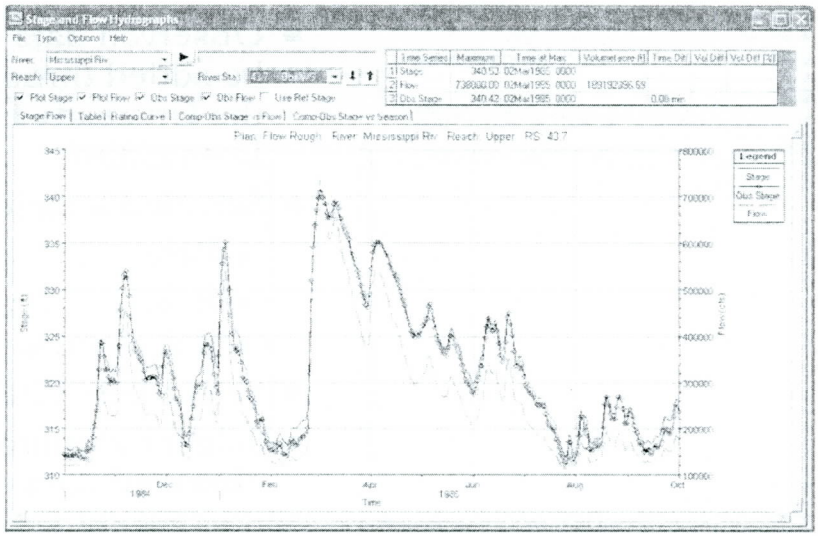
- Observed data can be added through the Unsteady Flow Data Editor
- Selected location
  - River
  - Reach
  - River station
- Select DSS File
- Select pathname for observed data

24

Observed 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 to table". 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. Observed data can be compared to simulated results for calibration. The observed data must be in DSS, and paths are associated with cross sections with the "Observe Data in DSS" dialog. After the connections have been set, the observed flow and/or stage data will automatically be plotted with the simulated flow and/or stage data on the "Flow and Stage Plotter". Observed stages will also appear on the profile and cross section plots (HEC).

Observed Data – Example Flow and Stage Plot

WEST
CONSULTANTS, INC
www.westconsultants.com



25

WEST

CONSULTANTS, INC.
www.westconsultants.com

Model Stability, Accuracy, and Sensitivity

- **Objective**: This lecture will discuss the components to creating a stable and accurate RAS unsteady flow model

This lecture was originally developed by the staff and engineers at the US Army Corps of Engineers Hydrologic Engineering Center (HEC).

1

Topics to Discuss

WEST CONSULTANTS, INC.
www.westconsultants.com

- Model accuracy
- Model stability
- Factors affecting accuracy and stability
  - Cross section spacing
  - Computational time step selection
  - Other parameters
- Common stability problems
- Detecting stability problems
- Model sensitivity
- Steps for creating a stable model

2

Model Accuracy

WEST CONSULTANTS, INC.
www.westconsultants.com

- Accuracy can be defined as the degree of closeness of the numerical solution to the true solution
- Accuracy depends upon the following:
  - Assumptions and limitations of the model (e.g., one dimensional model)
  - Accuracy of the geometric data (cross sections, Manning's 'n' values, bridges, culverts, etc.)
  - Accuracy of the flow data and boundary conditions
  - Numerical accuracy of the solution scheme

3

Numerical Accuracy

WEST CONSULTANTS, INC.
www.westconsultants.com

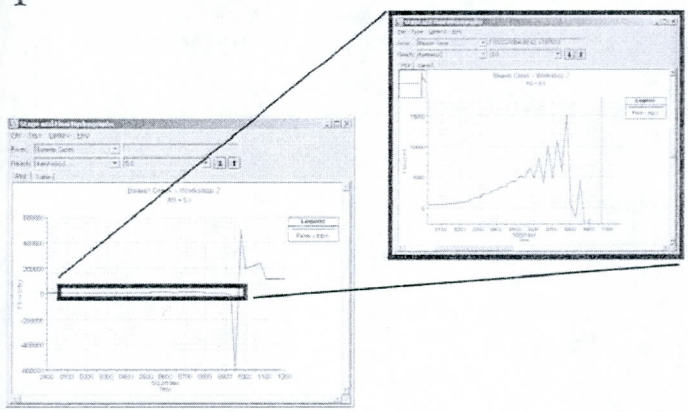
- If we assume that the one-dimensional unsteady flow equations are a true representation of flow moving through a river system, then only an analytical solution of these equations will yield an exact solution
- An exact solution of the equations is not feasible for complex river systems, so HEC-RAS uses a finite difference scheme
- Finite difference solutions are approximate

4

Model Stability

WEST CONSULTANTS, INC.
www.westconsultants.com

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



5

Factors Affecting Model Stability and Numerical Accuracy

WEST CONSULTANTS, INC.
www.westconsultants.com

- Cross section spacing
- Computation time step
- Theta weighting factor
- Solution tolerances and number of iterations
- Structure and spillway stability factors
- Structure and spillway exponential decay factors

6

Cross Section Spacing

WEST CONSULTANTS, INC. www.westconsultants.com

- Cross sections should be placed at representative locations to describe the changes in geometry
  - Samuels (1989) and Fread (2003) have equations for maximum spacing
  - Other references (USGS, USACE)
- Additional cross sections should be added at locations where changes occur in discharge, slope, velocity, and roughness
- Cross sections must also be added at levees, bridges, culverts, and other structures

7

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

Other references that discuss cross-section spacing:

Davidian, J. (1984). "Computation of water-surface profiles in open channels." USGS Techniques of Water-Resources Investigations, Book 3, Chapter A15. http://water.usgs.gov/pubs/twri/twri3-a15/

Thompson, D.B., and Rogers, T.D. (1993). "Water surface profile computations – how many sections do I need?" Proc., ASCE Conference on Hydraulic Engineering, San Francisco, CA, July 25-30, 791-796.

USACE. (1986). "Accuracy of computed water surface profiles." Hydrologic Engineering Center, RD 26, Davis, CA.

Samuels' Equation for Cross Section Spacing

WEST CONSULTANTS, INC. www.westconsultants.com

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

Where: Δx is the cross section spacing distance;
D is the bankfull depth (m); and
S_0 is the bed slope

8

Fread's Equation for Cross Section Spacing

WEST CONSULTANTS, INC.
www.westconsultants.com

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

Where: Δx is the cross section spacing distance;
c is the wave celerity; and
T_r is minimum rise time of the hydrograph (sec.)

9

Cross Section Spacing - Slope

WEST CONSULTANTS, INC. www.westconsultants.com

- Bed slope plays an important role in cross section spacing
  - Steeper slopes require more cross sections
  - Streams flowing at high velocities may require cross sections on the order of 100 feet or less
  - Larger uniform rivers with flat slopes may only require cross sections on the order of 1000 ft or more

10

Cross Section Spacing – How do you know if you have enough XS?

WEST CONSULTANTS, INC. www.westconsultants.com

- Use the HEC-RAS cross section interpolation
- Make a new plan and run the model
- Compare the before and after
- If no significant difference, then OK!

11

Computational Time Step

WEST CONSULTANTS, INC.
www.westconsultants.com

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

$$C_r = V_w \frac{\Delta t}{\Delta x} \leq 1.0 \qquad V_w = \left|\overline{V} \pm c\right| \qquad \Delta t \leq \frac{\Delta x}{V_w}$$

Where: V_w = The flood wave speed
V = Average velocity of the flow
c = wave celerity $= \sqrt{gH_d}$
Δx = Distance between cross sections
Δt = computational time step

- However, you may be able to run with larger time steps
  - Courant Condition is for explicit models
  - RAS unsteady is an implicit model

12

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

V = Average velocity of the flow

c = wave celerity

Δx = Distance between cross sections

Δt = computational time step

H_d = Hydraulic Depth, area/top width

g = gravitational acceleration

Computational Time Step *(Continued)*

WEST CONSULTANTS, INC. www.westconsultants.com

Wave celerity: $c = \sqrt{gH_d}$

Flood wave velocity: $V_w = \overline{V} + \sqrt{gH_d}$

For large rivers, the flood wave velocity is calculated more accurately by: $V_w = \dfrac{dQ}{dA}$

An approximate flood wave velocity can be calculated as: $V_w = \dfrac{3}{2}\overline{V}$

13

Where:

V_w = Flood Wave Speed

H_d = Hydraulic Depth, area/topwidth

Q = flow rate

A = Flow area

g = gravitational acceleration

Practical Time Step Selection

WEST CONSULTANTS, INC.
www.westconsultants.com

- For medium to large rivers the Courant condition may yield time steps that are too restrictive
  - A larger time step could be used and still maintain accuracy and stability
- A practical time step is = $\Delta t \leq \frac{T_r}{20}$

T_r = time to rise of the floodwave

- However, you will probably need to use a smaller time step when you have lateral structures/spillways and hydraulic connections between storage areas and the river system

14

A general rule of thumb is to use a computational interval that is equal to or less than the time of rise of the floodwave divided by 24 (RAS User's Manual, Ver. 3, January 2001)

Fread's Equation for Time Step Selection

WEST CONSULTANTS, INC.
www.westconsultants.com

$$\Delta t = \frac{T_r}{M} \qquad M \approx 2.67\left[1 + \frac{\mu' n^{0.9}}{q^{0.1} S_0^{0.45}}\right]$$

Where: Δt is the computational time step;
T_r is minimum rise time of the hydrograph (sec.);
μ' is 3.97 (3.13 for SI units);
n is the Manning friction coefficient;
q is the peak flow per unit width; and
S_0 is the bed slope

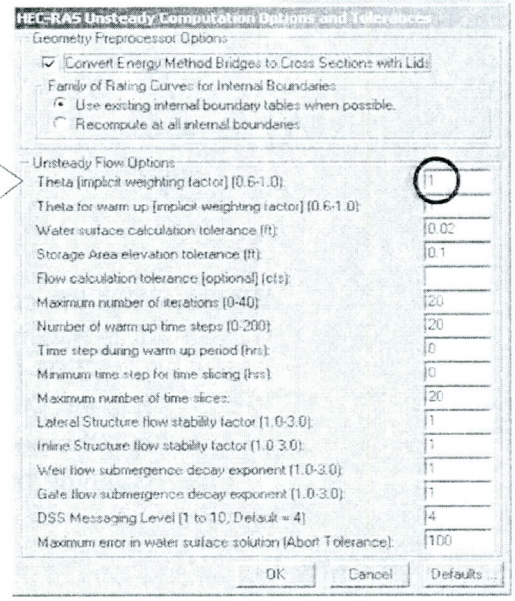
$6 < M < 40$
Typically, M is taken to be 20 (see previous slide)

15

Theta Weighting Factor

WEST CONSULTANTS, INC.
www.westconsultants.com

- A weighting applied to the finite difference approximations
- Theoretically can vary from 0.5 to 1.0
  - Practical limit is from 0.6 to 1.0
- 1.0 = most stable
- 0.6 = most accurate
- The default in RAS is 1.0
  - Reduce towards 0.6 as long as the model stays stable



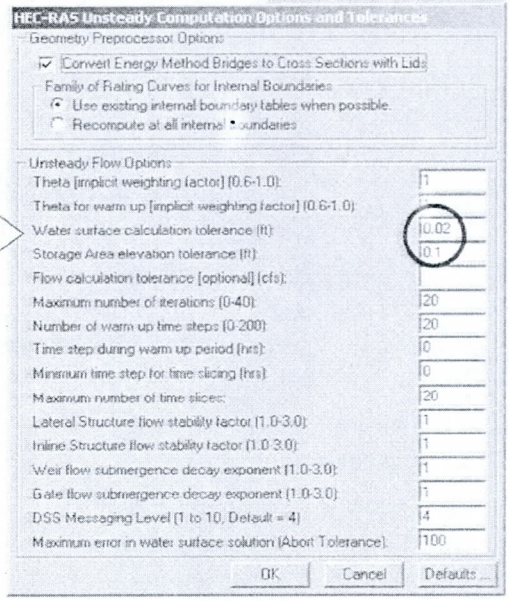
16

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 (HEC).

Solution Tolerances

WEST CONSULTANTS, INC.
www.westconsultants.com

- Two solution tolerances can be edited by the user:
  - Water surface calculation (default is 0.02 feet)
  - Storage area elevation (default is 0.05 feet)
- The default values should be good for most river systems - change them if you are sure!!!
  - Larger tolerances = potentially less accuracy and reduction in the stability of the solution
  - Smaller tolerance = potentially running to the maximum number of iterations every time step

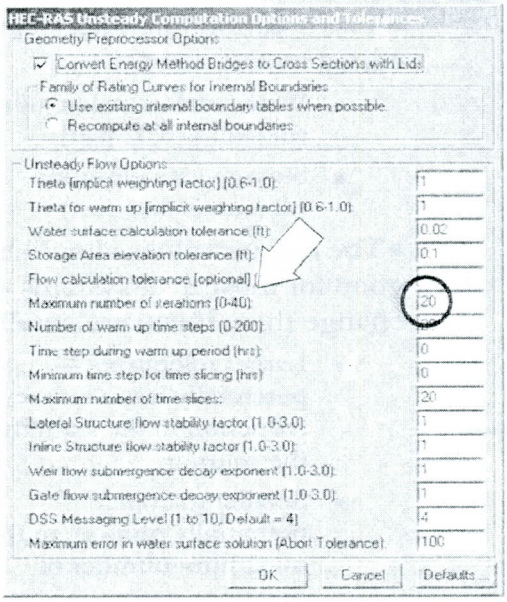


17

Solution Iterations

WEST CONSULTANTS, INC.
www.westconsultants.com

- At each time step:
  - Derivatives are estimated
  - Equations solved
  - Computation nodes checked for numerical error
  - RAS will continue to iterate if the error is greater than the allowable tolerances
- The default number of iterations is 20
- More iterations generally improve the solution
  - Especially true for lateral structures & storage areas

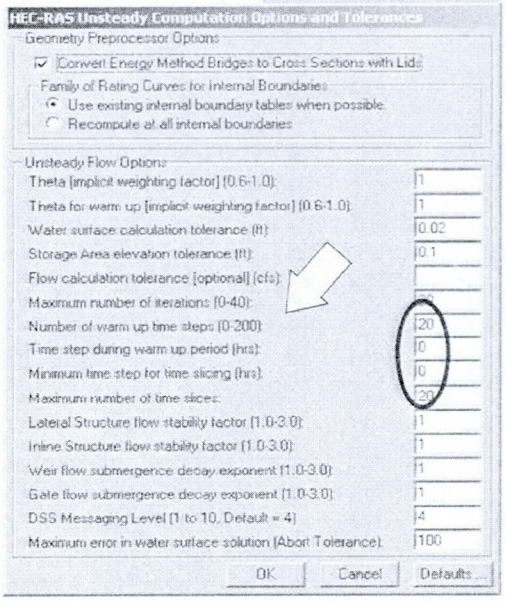


18

Warm Up Period and Time Slicing

WEST CONSULTANTS, INC.
www.westconsultants.com

- Maximum number of warm up time steps
  - RAS computes results for a constant inflow before the simulation begins
  - Creates a more stable solution at the beginning of the simulation
- Warm up time step
  - A specific time step for the warm up period
- Time slicing
  - Time steps can be automatically changed for very steep hydrographs
  - Used with the "Time Step Adjustment Options" in the Unsteady Flow Editor



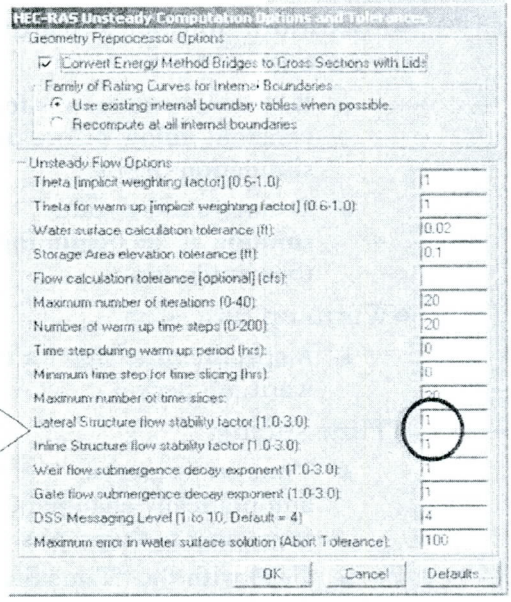
19

Maximum number of warm-up time steps: Before the dynamic simulation, the program runs a series of time steps with constant inflows. This is called a warm up period. This is done in order to smooth the profile before allowing the inflow hydrographs to progress. This helps to make a more stable solution at the beginning of the simulation. The default number of warm up time steps is set to 20. This value ranges from 0 to 40. **Time step during warm up period:** During the warm up period described in the previously, it is sometimes necessary to use a smaller time step than what will be used during the unsteady flow calculations. The initial conditions from the backwater analysis uses a flow distribution in the reaches which is often different than that computed by unsteady flow. This can cause some instabilities at the beginning of the simulation. The use of a smaller time step during the warm up period helps to get through these instabilities. The default is to leave this field blank, which means to use a time step that is the same as for the unsteady flow simulation period. **Minimum time step for time slicing:** The program has an option to interpolate between time steps when it finds a very steep rise in an inflow hydrograph, or a rapid change in stage at any cross section. This option allows the user to set a minimum time step to use during interpolation. This prevents the program from using to small of a time step during time slicing. **Maximum number of time slices:** This option defines the maximum number of interpolated time steps that the program can use during time slicing, as described in the previous paragraph (RAS User's Manual, Ver. 3, January 2001, p. 8-23).

Structure and Spillway Stability Factors

WEST CONSULTANTS, INC.
www.westconsultants.com

- Lateral and Inline 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 structure/spillway is assumed to be constant
  - A large change in flow during a time step may cause oscillations
  - First consider reducing the time step
- Lateral and Inline Structure stability factors can improve stability
  - Range of allowable values = 1.0 to 3.0
  - 1.0 = most accurate = least stable
  - 3.0 = least accurate = more stable



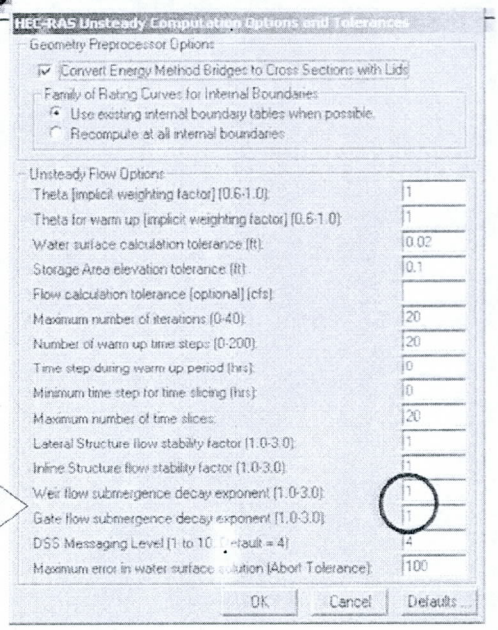
20

Structure flow stability factor: This factor is used to increase the stability of the numerical solution in and around a structure. The factor varies from 1.0 to 3.0. As the value is increased, the solution is more stable but less accurate. A value of 1.0 is the most accurate, but is susceptible to oscillation in the computed structure flow. (RAS User's Manual, Ver. 3.0, January 2001, p. 8-24)

Structure and Spillway Exponential Decay Factors

WEST CONSULTANTS, INC.
www.westconsultants.com

- For highly submerged structures
- Oscillations from large flow changes are caused by the combination of:
  - Steep curve in discharge reduction (at 95% to 100% submergence ratio)
  - Flow in RAS must be greater than zero over a submerged structure
- The decay factors dampen the flow across the structure
  - Range = 1.0 to 3.0
  - 1.0 = no effect
  - 3.0 = greatest dampening

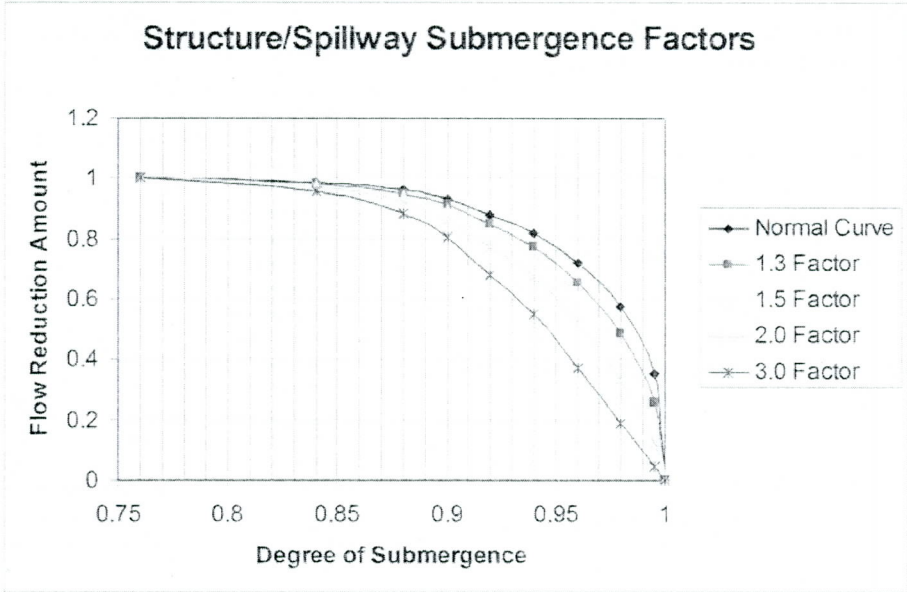


21

When you have a structure or spillway connecting two storage areas, or a storage area and a reach, oscillations can occur when the structure or spillway becomes highly submerged. The program must always have flow going one way or the other when the water surface is above the structure/spillway. When a structure/spillway is highly submerged, the amount of flow can vary greatly with small changes in stage on one side or the other. This is due to the fact that the submergence curves, which are used to reduce the flow as it becomes more submerged, are very steep in the range of 95 to 100 percent submergence. The net effect of this is that you can get oscillations in the flow and stage hydrograph when you get to very high submergence levels. The program will calculate a flow in one direction at one time step, that flow may increase the stage on the receiving side of the structure, so the next time step it sends flow in the other direction. This type of oscillation is ok if it is small in magnitude. However, if the oscillations grow, they can cause the program to go unstable (March 2002 Course Lecture Notes).

Structure/Spillway Submergence Exponents

WEST CONSULTANTS, INC. www.westconsultants.com



22

To reduce the oscillations, the user can increase the Structure/Spillway Submergence Factor. 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 (HEC March 2002 Lecture Notes). The theory from the "normal" curve (for a factor of 1) is based on "Hydraulics of Bridge Waterways" (Bradley, 1978) as discussed in Chapter 5, Modeling Bridges, of the RAS Technical Reference. Using a factor other than 1 can improve stability but diverges from the original theory (WEST 06.10.02).

Common Stability Problems

WEST CONSULTANTS, INC. www.westconsultants.com

- Too large of a time step
- Not enough cross sections
- Model goes to critical depth
  - Use the mixed flow feature
- Abrupt changes in downstream boundary condition
  - Ill-defined rating curve
  - Steep slope for normal depth
- Abrupt changes in cross section properties
  - Revise levee or ineffective flow definition

23

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 given cross section are changing rapidly with respect to time, the program will go unstable. The solution to this problem is to decrease the time step. **Not enough cross sections**: When cross sections are spaced far apart, and the changes in hydraulic properties are great, the solution can become unstable. **Model goes to critical depth**: 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 temporary 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. (For Beta Version 3.1: The default solution method for unsteady flow routing within HEC-RAS is generally for subcritcal flow. 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). **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 stages that are too low; and normal depth boundaries where the user has entered too steep of a slope. **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 (HEC March 2002 Lecture Notes).

Common Stability Problems (Continued)

WEST CONSULTANTS, INC. www.westconsultants.com

- Ill defined hydraulic properties tables
  - Elevation not high enough
  - Not enough definition
- Abrupt transitions in bridge/culvert family of rating curves
- Wide and flat lateral structures/spillways
  - Too much flow overtopping flow in a time step.
- Gated spillways opening/closing too quickly
- Large influx of water to a shallow depth channel

24

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. **Bad 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 = 40), how many points in each curve (default = 10), 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. **Wide and flat Lateral structures/Spillways**: during the computations there will be a point at which for one time step no flow is going over the lateral structure, and then the very next time step there is. If the water surface is rising rapidly, and the structure is wide and flat, the first time the water surface goes above the structure could result in a very large flow being computed (i.e. it does not take a large depth above the structure 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 structures between storage areas. The solution to this problem is to use smaller computational time steps, and/or structure/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. **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 (HEC March 2002 Lecture Notes).

Detecting Stability Problems

WEST CONSULTANTS, INC.
www.westconsultants.com

- How do you know you have a stability problem?
  - Program completely blows up during run
  - Program goes to maximum number of iterations for several time steps in a row
    - This does not always indicate stability problems
  - Program has oscillations in the computed stage and/or flow hydrographs

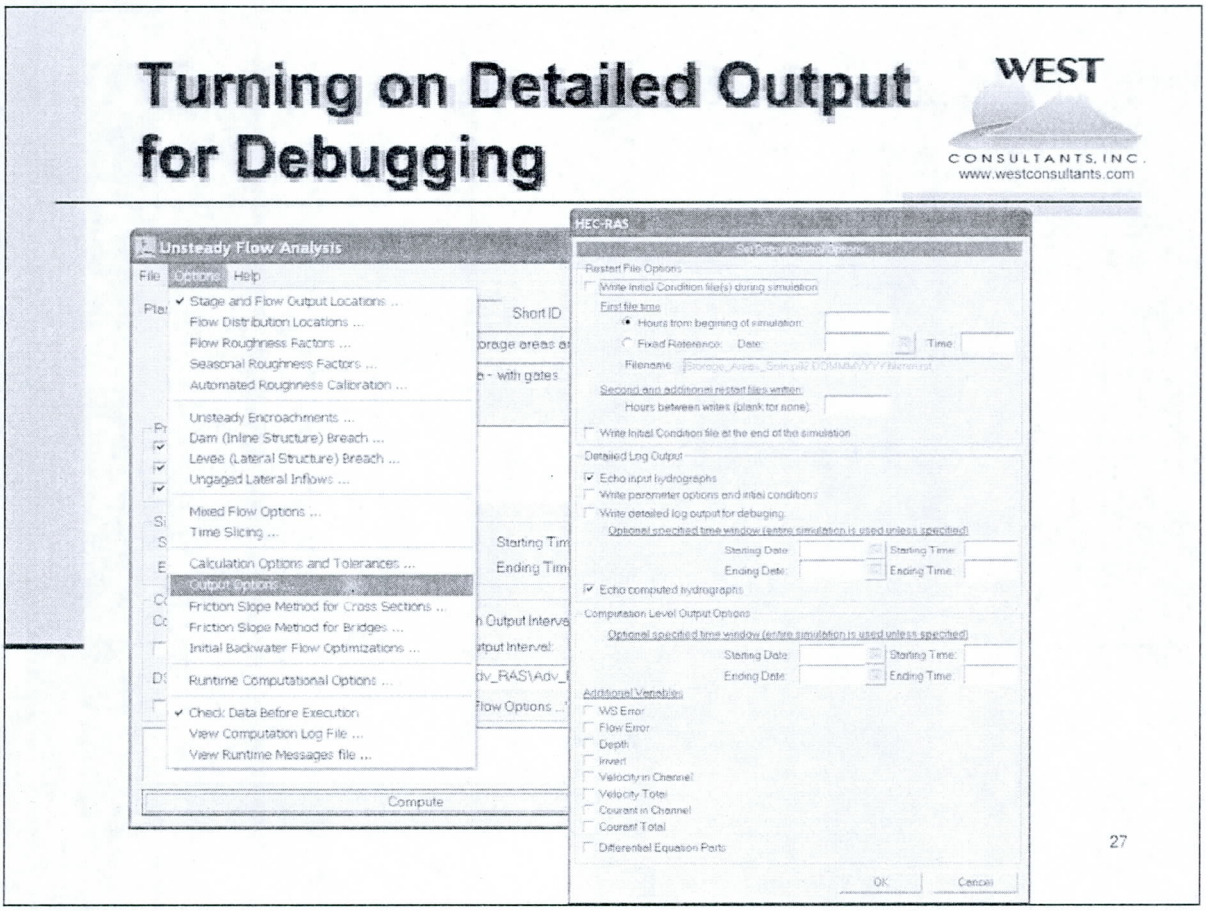
25

Identifying Stability Problems

WEST CONSULTANTS, INC.
www.westconsultants.com

- Note the simulation time when the program either blew up or first started to oscillate
- 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
- Check the data in this general area

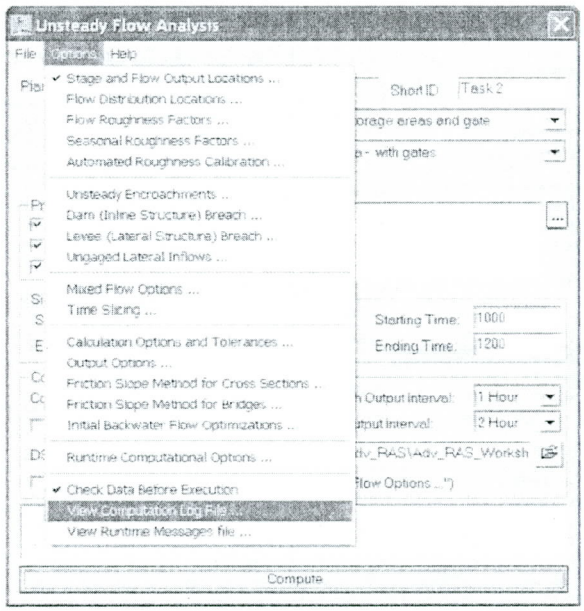
26



As shown in 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 becomes unstable or will not converge on a solution. 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 (HEC March 2002 Lecture Notes).

Viewing Detailed Log Output

WEST CONSULTANTS, INC.
www.westconsultants.com

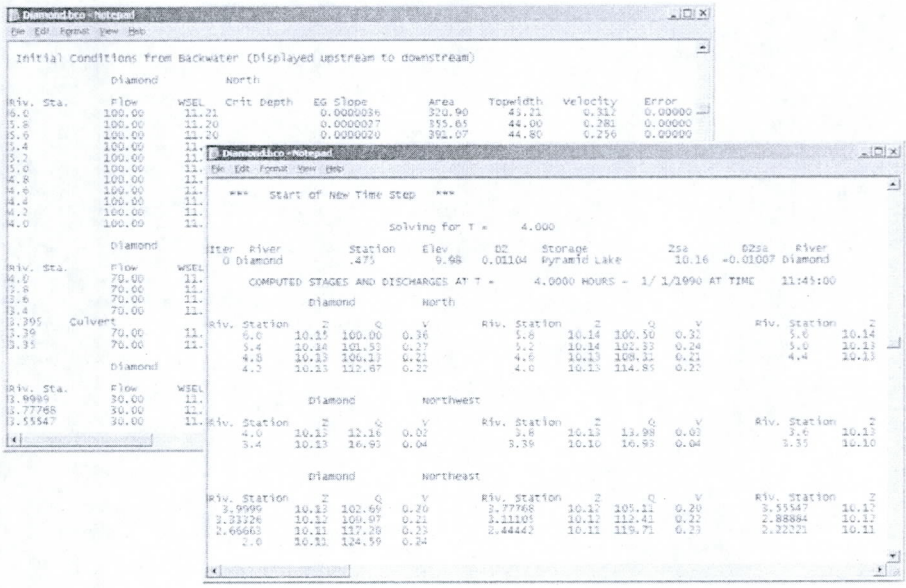


28

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) (HEC March 2002 Lecture Notes).

Example Detailed Log Output

WEST CONSULTANTS, INC.
www.westconsultants.com



29

The program lists the computed initial conditions from a backwater calculation for each of the river/reaches. They are listed in the order they were computed during the backwater analysis which is downstream to upstream.

The key to the column headers is presented below:

Iter	= Iteration Number
Station	= River station with the larges error in the calculated stage. River Stationing is by a code: reach number+ "S" + river stationing (example: 1S5.000 = reach 1, river station 5.000)
ELEV	= Computed water surface elevation at that river station
DZ	= The "Numerical Error" in the computed stage at that location
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
Cell #	= Number code for a storage area.
Zsa	= Computed elevation of the storage area
Dzsa	= The "Numerical Error" in the computed storage area elevation.

Example Problems Shown in Detailed Output

WEST CONSULTANTS, INC.
www.westconsultants.com

```
File Edit Format View Help

   ***   Start of New Time Step   ***

WARNING!  Extrapolated above the top of the property table at XSEC(5):
AT   R.S. 5.0      W.S. =     215.76                                  in River/Reach Beaver Creek     Kentwood

                          Solving for T =    24.900

Iter  River          Station    Elev        DZ      Storage          Zsa        DZsa     River          Station
   0 Beaver Creek     5.0       210.64    3.07246   Sto #1          205.93   0.01684 Beaver Creek     5.0        3
   1 Beaver Creek     5.0       209.46   -1.75517   Sto #1          205.93   0.00000 Beaver Creek     5.0        3
   2 Beaver Creek     5.0       209.50   -1.37187   Sto #1          205.93   0.00000 Beaver Creek     5.0        3
   3 Beaver Creek     5.0       207.63   -1.24392   Sto #1          205.93   0.00000 Beaver Creek     5.0        3
   4 Beaver Creek     5.0       206.61   -1.44413   Sto #1          205.93   0.00000 Beaver Creek     5.0        3
   5 Beaver Creek     5.0       205.08   -3.18797   Sto #1          205.93   0.00000 Beaver Creek     5.0        3
   6 Beaver Creek     5.0       201.36   -5.27021   Sto #1          205.93   0.00000 Beaver Creek     5.0        3
   7 Beaver Creek     5.0       199.91   18.23091   Sto #1          205.93   0.00000 Beaver Creek     5.0
   8 Beaver Creek     5.0       199.91    7.90089   Sto #1          205.93   0.00000 Beaver Creek     5.016
   9 Beaver Creek     5.0       199.91    8.62846   Sto #1          205.93   0.00000 Beaver Creek     5.016
  10 Beaver Creek     5.0       199.91   11.98420   Sto #1          205.93   0.00000 Beaver Creek     5.0        3
  11 Beaver Creek     5.0       199.91    3.29980   Sto #1          205.93   0.00000 Beaver Creek     5.0        3
  12 Beaver Creek     5.0       199.91   66.70975   Sto #1          205.93   0.00000 Beaver Creek     5.0        3

WARNING!  Extrapolated above the top of the property table at XSEC(5):
From R.S. 5.048    W.S. =     214.10  to R.S. 5.016    W.S. =     219.01  in River/Reach Beaver Creek     Kentwood

  13 Beaver Creek     5.0       199.91    5.92217   Sto #1          205.93  -0.00001 Beaver Creek     5.0

WARNING!  Extrapolated above the top of the property table at XSEC(5):
AT   R.S. 5.016    W.S. =     215.28                                  in River/Reach Beaver Creek     Kentwood

  14 Beaver Creek     5.0       199.91    7.67301   Sto #1          205.93   0.00000 Beaver Creek     5.0        3
  15 Beaver Creek     5.0       199.91    3.83929   Sto #1          205.93   0.00000 Beaver Creek     5.0        3
  16 Beaver Creek     5.0       199.91   13.05661   Sto #1          205.93   0.00000 Beaver Creek     5.0
  17 Beaver Creek     5.0       199.91    8.04358   Sto #1          205.93   0.00000 Beaver Creek     5.0        3
  18 Beaver Creek     5.0       199.91    8.83572   Sto #1          205.93   0.00000 Beaver Creek     5.0        3
  19 Beaver Creek     5.0       199.91   11.67928   Sto #1          205.93   0.00000 Beaver Creek     5.0        3
  20 Beaver Creek     5.0       199.91    5.55514   Sto #1          205.93   0.00000 Beaver Creek     5.0        3

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

      COMPUTED STAGES AND DISCHARGES AT T =     24.9000 HOURS -  2/21/1999 AT TIME     0:53:00

            Beaver Creek     Kentwood

Riv. Station     T        Q        V        Riv. Station     T        Q        V        Riv. Station     T        Q        V
   5.99    219.55  13948.   2.41             5.9     218.07  13956.   2.41             5.42    213.90  13977.   4.75
   5.31    215.40   7884.   2.18             5.274   211.28   7891.   2.08             5.0     207.63  38240.   3.57
```

30

If the program goes to the maximum number of iterations, it will choose the iteration that had the minimum amount of error, and then go to the next time step.

Note: do a search for the word **"Warning"** to quickly get to the area of potential problems. (HEC)

Fixing an Unstable Unsteady Flow Model

WEST CONSULTANTS, INC. www.westconsultants.com

- Use steps suggested for creating a stable model
- Verify that HTab values are not exceeded
- Consider adding a pilot channel for reaches with low flow
- Problems with lateral structures?
  - Initially raise structure elevations above the maximum peak stage
  - Lower elevations until the root of the problem is identified
- Problems with bridges?
  - Verify smooth transitions in the rating curve tables
  - Temporarily change default minimum structure flow and maximum low cord elevations to help identify the problem
- You may have to start with a "flooded" system, then lowering the stage to the initial conditions for your modeling scenario

31

Model Sensitivity

WEST CONSULTANTS, INC. www.westconsultants.com

- Numerical sensitivity:
  - Computation time step
    - Do results change significantly with a smaller value?
  - Theta
    - Reduce towards 0.6
  - Structure/spillway stability factors
    - Reduce towards 1.0
  - Structure/spillway exponential decay factors
    - Reduce towards 1.0

32

Computational Time Step - The user should try a smaller time step to see if the results change significantly. If the results do change significantly, then the original time step is probably too large to solve the problem accurately. **Theta Weighting Factor** - The default value for this factor is 1.0, which provides the greatest amount of stability for the solution, but may reduce the accuracy. After the user has a working model, this factor should be reduced towards 0.6 to see if the results change. If the results do change, then the new value should be used, as long as the model stays stable. Be aware that using a value of 0.6 gives the greatest accuracy in the solution of the equations, but it may open the solution up to stability problems.
Structure/Spillway Stability Factors – If you are using these factors to maintain stability, try to reduce them to the lowest value you can and still maintain stability. The default value is 1.0, which is no stability damping. **Structure/Spillway Submergence Exponents** – In general these parameters will not affect the answers significantly, they only provide greater stability when a spillway/structure as at a very high submergence. Try reducing them towards 1.0 (which is no factor) to see if the model will remain stable. (HEC)

Model Sensitivity - Continued

WEST CONSULTANTS, INC.
www.westconsultants.com

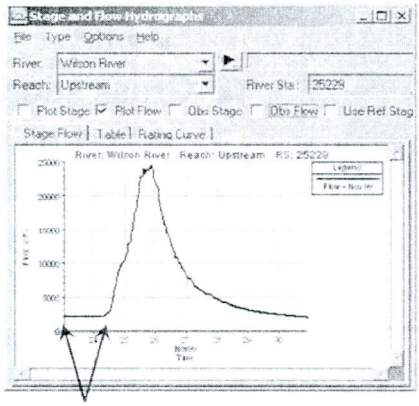
- Physical parameter sensitivity:
  - Manning's n values
    - ±10%
  - Cross section spacing
    - Test by interpolating
  - Cross section storage
    - Adjust ineffective flow areas
  - Structure/spillway coefficients
    - Lateral structures/spillway coefficients can have a great impact
  - Bridge/culvert parameters
    - Normally only effect the locally computed stages, unless it is a flat area in which the bridge causes great backwater

33

Suggested Steps for Creating a Stable Unsteady Flow Model

WEST CONSULTANTS, INC.
www.westconsultants.com

1. Start simple
   - Create the main stem reach
   - Leave out structures (e.g., bridges, lateral structures, etc.)
   - Run in unsteady mode with a constant inflow hydrograph
   - Not running?
     - Check your initial flow distribution
     - Are HTab values exceeded?



Add steady flow to boundary hydrograph if necessary

34

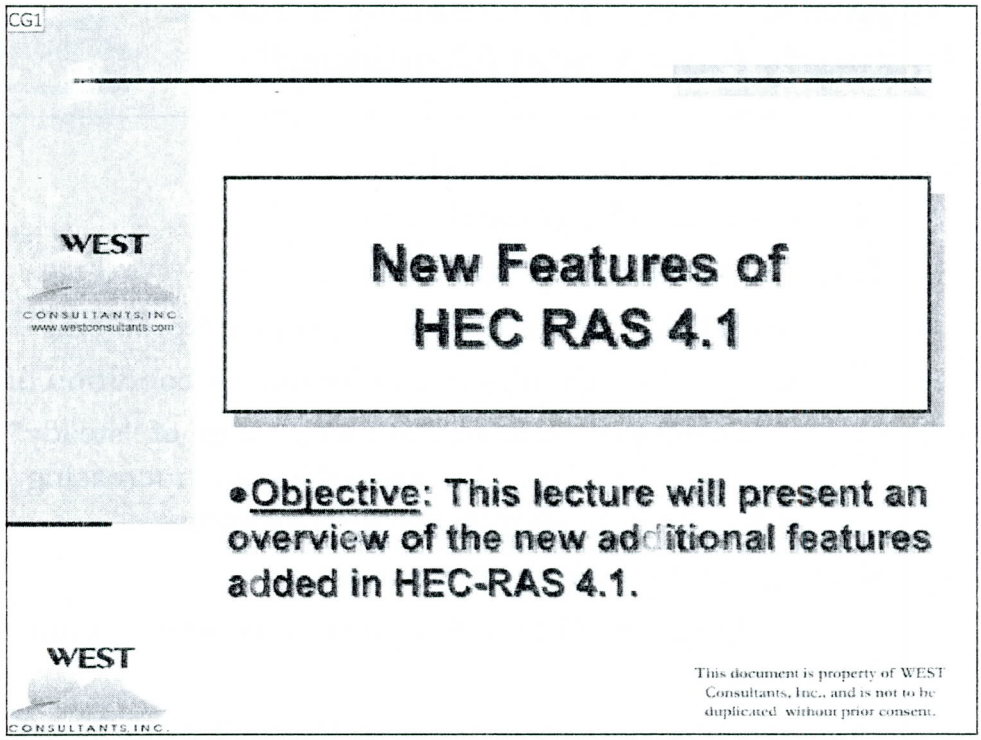
Using the initial unsteady flow value as for your steady flow will not only help to create an initial stable model, but it also will assist in determining the initial flow values. Iterate on initial stage values if trouble exists solving for the initial solution – compare to the computed initial values in the results table or log files.

Suggested Steps for Creating a Stable Unsteady Flow Model *(Continued)*

WEST CONSULTANTS, INC. www.westconsultants.com

2. Add in any additional reaches
   - Side channels, tributaries, etc.
3. Include the unsteady flow data
   - Fix HTab, ineffective flow, or geometry problems
   - Are there instabilities in your boundary condition data?
   - May need to run the model with a range of "steady" unsteady flow files, or a gradual linearly increasing hydrograph, to identify problem locations
4. Systematically include structures
   - Fix HTab, ineffective flow area, or structure geometry problems
   - Ensure the model runs successfully after each structure is added

35



This lecture was initially written by Brian Wahlin, Ph.D., P.E., D.WRE of WEST Consultants, Inc. in September, 2010. Further contributions have been made to this lecture by the staff of WEST Consultants.

Release History

WEST CONSULTANTS, INC.
www.westconsultants.com

- Version 3.0.1 (released March 2001)
  - Unsteady flow
  - Lateral structures (e.g., levees)
- Version 3.1 (released November 2002)
  - Unsteady encroachment
  - Navigation dams
  - Structures
    - Pump stations
    - Dam break/levee breach
- Version 3.1.1 (released May 2003), Version 3.1.2 (April 2004) and Version 3.1.3 (released May 2005)
  - Bug fixes
- Version 4.0.0 (beta released December 2006)
  - Sediment transport
  - Sediment Impact Analysis Methods (SIAM)
  - Temperature modeling
  - Pressurized pipe flow

2

Release History

WEST CONSULTANTS, INC. www.westconsultants.com

- Version 4.0.0 (released March 2008)
  - Channel design/modification tools
  - Added gate types
    - overflow gates w/ closed or open top
  - User defined rules for controlling gate operations
  - Added lateral structure functionality
  - Geo-referencing tools
  - Additional graphical outputs
  - Shortcut keys to graphics
  - Pump station override rules
  - Hager's lateral weir equation
  - Users manual and help system
- Version 4.1.0 (released January 2010)

•Channel Design/Mod tools - Available in the tools menu of the Geometric Data Editor. Allows user to make trapezoidal cuts in existing geometry or create new geometry.

•New Gate types - Overflow gates with open or closed top, also allows user to define a curve to represent the gate.

•Gate Operation Rules - The user can define current flows, water surfaces, and stages at and near structures and the time of day and time of year.

•Lateral structure functionality - The user can set flow to be distributed across multiple cross-sections along a lateral weir and can set spacing between cross sections on the weir.

•Geo-referencing - Found in the GIS tools menu. Allows the user to edit x and y coordinates in table format.

•Graphical outputs - For sediment transport, SIAM analyses, and water quality computations. Also allows the user to view unsteady flow output at the computation interval level.

•Shortcut keys - In graphic window hold "shift" to pan and "ctrl" for a measuring tool.

•Pump station. The user can turn pumps on and off based on time of day and target flows and stages. Rules can also be set to override pumps max and min flow capacity.

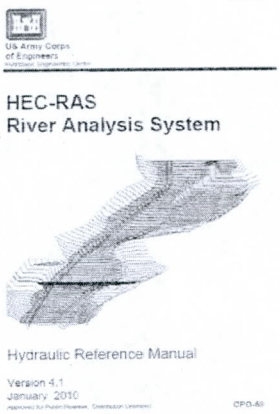
•Hagar' equation - Weir discharge coefficient is computed automatically based on physical and hydraulic properties.

•Users Manual - Completely updated for version 4.0 including new chapters. New help system uses manuals PDF file.

All New Manuals

WEST
CONSULTANTS, INC
www.westconsultants.com

- Updated text and graphics
- Greatly expanded
- User's Manual
- Hydraulic Reference Manual
- Applications Guide
- Accessible via the "Help" item.



Bug Fixes

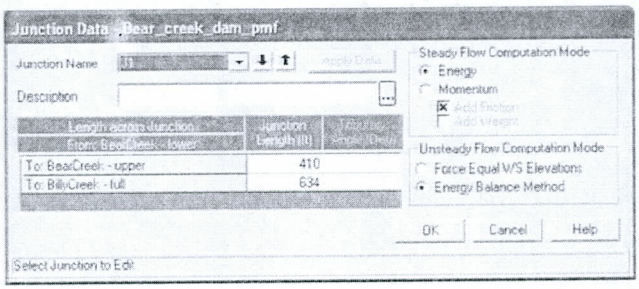
WEST CONSULTANTS, INC.
www.westconsultants.com

- 50 bug fixes in 4.1
- 60 bug fixes in 4.0 beta

Unsteady Junctions

WEST CONSULTANTS, INC.
www.westconsultants.com

- Problems with unsteady flow with steep slopes and shallow flow.
- Energy method now an option.



Unsteady Junctions

WEST CONSULTANTS, INC
www.westconsultants.com

- Normal combining junction
  - All cross-sections bounding junction are given same water surface elevation
- Normal split flow junction
  - Water surface elevations are based on the computed values upstream of junction
- Problems
  - Cross-sections need to be close to junction
  - Dependent on slope

Unsteady Junctions

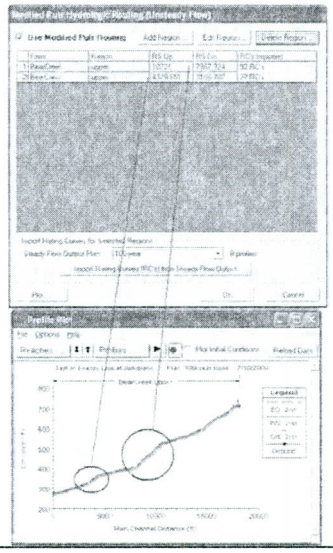
WEST CONSULTANTS, INC. www.westconsultants.com

- New Energy Balance Method
  - Water surfaces elevations are not forced to be the same
  - Energy balance across the junction
- Useful
  - Medium to steep streams
  - Long spacing between cross-section and junction

Hydrologic Routing Reaches

WEST CONSULTANTS, INC.
www.westconsultants.com

- Steep, shallow reaches seem to often cause problems
- Hydrologic Routing Reach
  - Modified Puls Hydrologic routing technique



Hydrologic Routing Reaches

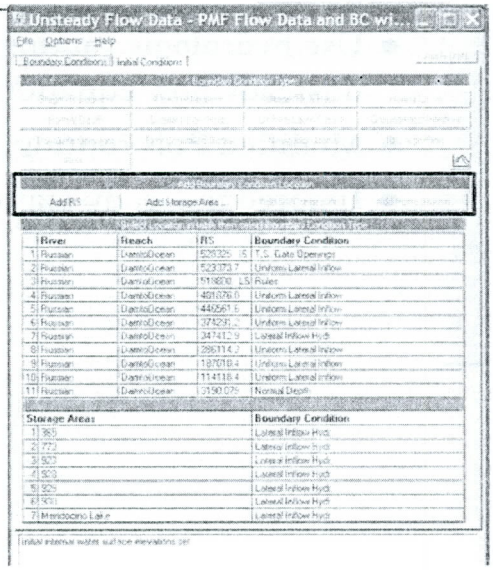
WEST CONSULTANTS, INC. www.westconsultants.com

- Use hydrologic routing techniques instead of unsteady flow equations
- Modified Puls routing
- Ignored in steady-state models

New Unsteady Flow Data and Boundary Conditions Editor

WEST CONSULTANTS, INC.
www.westconsultants.com

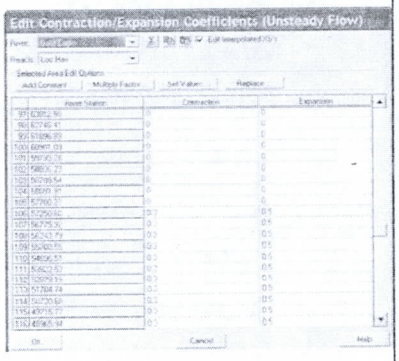
- Editors modified to be more flexible
- Attach more than one inflow hydrograph to a storage area
- Changes
  - Boundary conditions tab
  - How boundary conditions locations are established



Contraction/Expansion Losses for Unsteady Flow

WEST CONSULTANTS, INC.
www.westconsultants.com

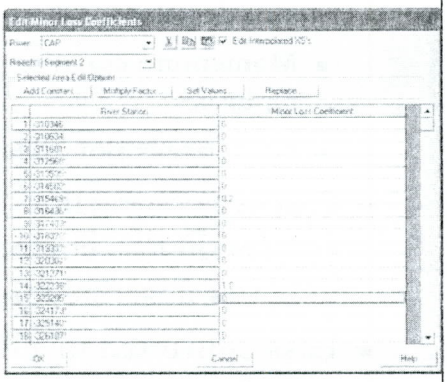
- Generally not used in unsteady flow
  - Momentum equation solves for losses
  - Sometimes not adequate
- Can use contraction / expansion loss coefficients to better describe forces
- Loss converted to a force and added to momentum equation



Minor Losses

WEST
CONSULTANTS, INC
www.westconsultants.com

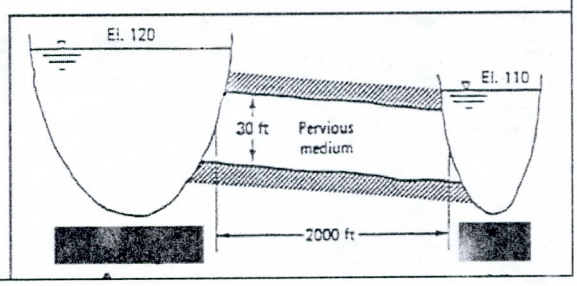
- Minor losses due to bends, junctions, etc.
- Loss coefficient K
- Steady flow
  - Minor loss added to the energy equation
- Unsteady flow
  - Minor loss converted to forced and added to momentum equation



Groundwater Leakage

WEST CONSULTANTS, INC. www.westconsultants.com

- Groundwater Interflow
  - River reaches
  - Now can be applied to storage areas
- Groundwater can enter storage area
- Surface water can leave storage area
- Darcy's equation



Water Quality Modeling Enhancements

WEST CONSULTANTS, INC
www.westconsultants.com

- Non-uniform channel geometry
- Steep drop
- Can lead to discontinuity and instabilities
- Dispersion coefficients
  - Individual water quality cell
  - Each time step
  - Function of velocity and channel geometry

Water Quality Modeling Enhancements

WEST CONSULTANTS, INC. www.westconsultants.com

- Constant temperature option
  - Overrides energy budget
  - Still need to enter all parameters needed for energy budget
- Supports reverse flow

Sediment Transport Modeling Enhancements

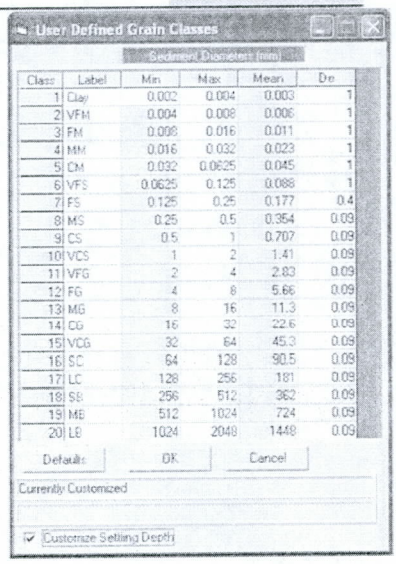
WEST CONSULTANTS, INC. www.westconsultants.com

- Calibration parameters for transport equations
- Wong and Parker correction to the MPM equation
- Pass through nodes
  - 100% of sediment that enters nodes passes through
  - No aggradation or degradation

Sediment Transport Modeling Enhancements

WEST CONSULTANTS, INC.
www.westconsultants.com

- Settling depth parameter can be edited by grain size
- Fractions were previously hard wired
- Could result in unrealistic settle depths

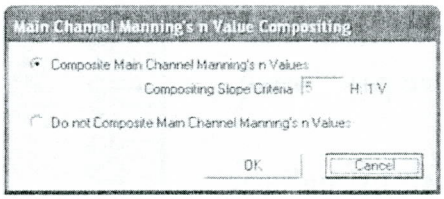


Class	Label	Min	Max	Mean	De
1	Clay	0.002	0.004	0.003	1
2	VFM	0.004	0.008	0.006	1
3	FM	0.008	0.016	0.011	1
4	MM	0.016	0.032	0.023	1
5	CM	0.032	0.0625	0.045	1
6	VFS	0.0625	0.125	0.088	1
7	FS	0.125	0.25	0.177	0.4
8	MS	0.25	0.5	0.354	0.09
9	CS	0.5	1	0.707	0.09
10	VCS	1	2	1.41	0.09
11	VFG	2	4	2.83	0.09
12	FG	4	8	5.66	0.09
13	MG	8	16	11.3	0.09
14	CG	16	32	22.6	0.09
15	VCG	32	64	45.3	0.09
16	SC	64	128	90.5	0.09
17	LC	128	256	181	0.09
18	SB	256	512	362	0.09
19	MB	512	1024	724	0.09
20	LB	1024	2048	1448	0.09

Manning's n Value Compositing

WEST CONSULTANTS, INC
www.westconsultants.com

- The user can change the Compositing Slope Criteria.



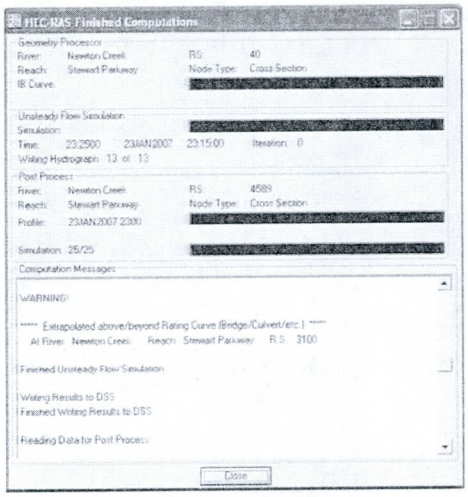
- Or a user can choose not to automatically composite the Manning's n values in a channel.

CG3

View Runtime Messages File

WEST CONSULTANTS, INC.
www.westconsultants.com

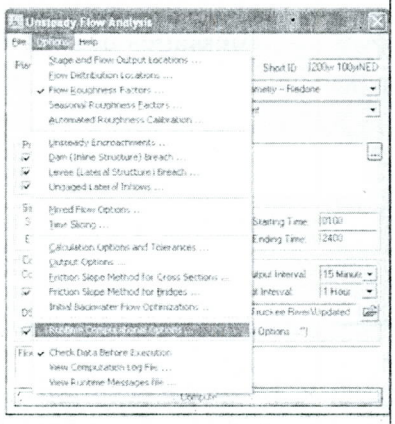
- RAS now stores runtime messages in a log file.

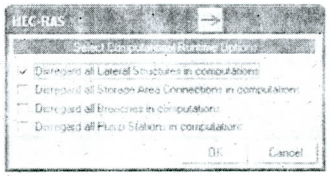


Runtime Computational Options

WEST CONSULTANTS, INC.
www.westconsultants.com

- During computations, disregard:
  - Lateral Structures
  - Storage Area Connections
  - Breaches
  - Pump Stations
- Great way to isolate problem areas.





In the Runtime Computational Options, you have the option to "disregard" Lateral Structures, Storage Area Connections, Breaches, and Pumps in the computations. Let's say you turn off lateral structures, rerun your model and it runs fine.

Then you know that there is a problem with one or more of your lateral structures. Simple diagnosing tool-but very effective.

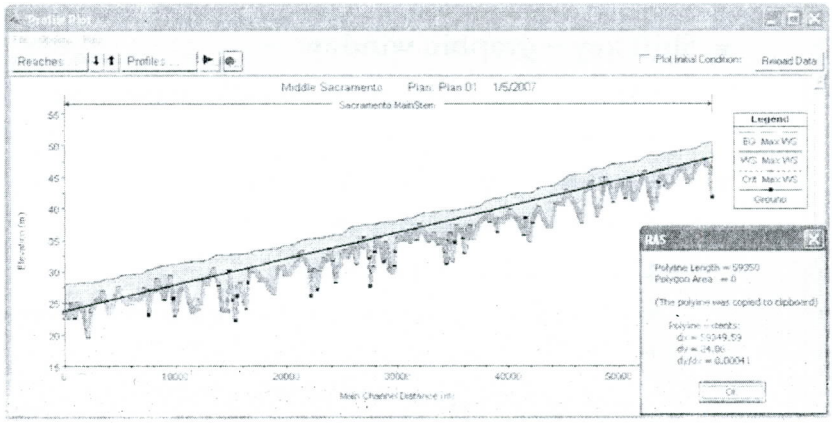
Not New, But Really Useful

WEST CONSULTANTS, INC. www.westconsultants.com

- Short Cut Keys
  - Shift key – graphic window
    - Mouse pointer changes to "hand" for panning mode
  - Control key – graphic window
    - Changes to measuring tool
    - Can create line or polygon
    - when released displays x, y distance traveled, slope of line or bounding box
    - x and y coordinates sent to clipboard

Not New, But Really Useful

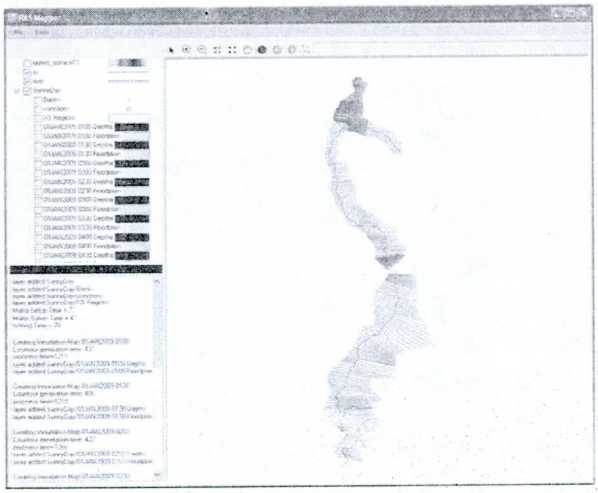
WEST
CONSULTANTS, INC
www.westconsultants.com



RAS Mapper

WEST CONSULTANTS, INC.
www.westconsultants.com

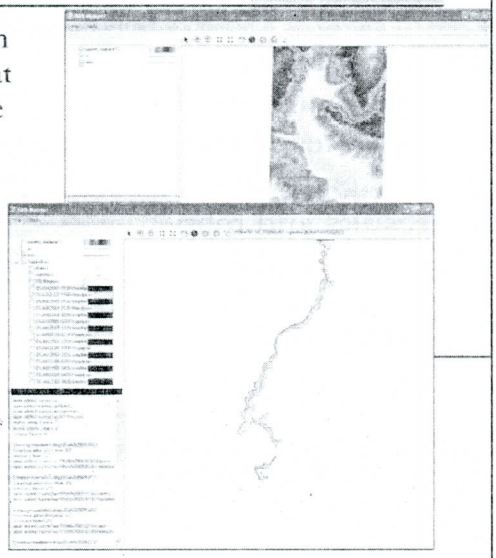
- RAS Mapper allows the user to take steady and unsteady flow results and map the floodplains directly in the HEC-RAS environment



RAS Mapper

WEST CONSULTANTS, INC.
www.westconsultants.com

- With the terrain model in floating point grid format (*.flt) the user can create a floodplain and associated depth layers.
- GIS is still required to create a geometry file, but the user is no longer required to go through the PostRAS procedure in GeoRAS



What's Next?

WEST CONSULTANTS, INC. www.westconsultants.com

- Enhance RAS Mapper capabilities
- Cut models inside Geometric Editor
- Integrate RAS and MODFLOW
  - OpenMI
- Automated roughness calibration
- Spreading model
  - Currently using FLO-2D
  - Write own code in the future

CG7

OpenMI (short for Open Model Integration Environment) is a data and model integration framework. The primary objective is to take otherwise independent data and computing systems, for example a database storing rainfall time series, a rainfall-runoff model, and a river hydraulics model, and provide a standard means for describing how time series are communicated between the systems. OpenMI was developed by HaromIT, which consists of a number European environmental software companies and university researchers. Software companies like DHI are working to make their models "OpenMI-compliant", thereby allowing environmental modelers the ability to couple different DHI models into modeling frameworks.

WEST
CONSULTANTS, INC.
www.westconsultants.com

HEC-RAS, River Analysis System

By WEST Consultants, Inc.

Basic Input

Data Requirements

Note: This lecture is based upon Chapter 3 of the HEC-RAS Hydraulic Reference Manual

This document is property of WEST Consultants, Inc., and is not to be duplicated without prior consent.

Overview

WEST
CONSULTANTS, INC.
www.westconsultants.com

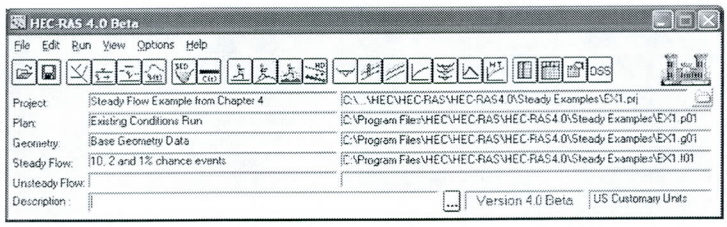
- General
- Geometric Data
  - Ground Points
  - Additional Data
- Optional Cross-section Data
- Junctions
- Steady Flow Data

2

General

WEST CONSULTANTS, INC.
www.westconsultants.com

- Overview: This lecture describes the basic data requirements for performing the one-dimensional flow calculations within HEC-RAS
- Main objective of HEC-RAS: Compute water surface elevations at all locations of interest for given flow values
- Data needed for computations:
  1. Geometric data
  2. Flow data
- HEC-RAS can perform both steady and unsteady flow computations; however, only steady flow is discussed in this class



3

Overview: This lecture describes the basic data requirements for performing the one-dimensional flow calculations within HEC-RAS

Main objective of HEC-RAS: Compute water surface elevations at all locations of interest for given flow values

Data needed for computations:

Geometric data

Flow data

Geometric data are required

HEC-RAS can perform both steady and unsteady flow computations

Only steady flow will be discussed in this section

Geometric Data

WEST CONSULTANTS, INC. www.westconsultants.com

- Discussed in this lecture:
  - The River System Schematic
  - Cross Section Geometry
  - Optional Cross Section Properties
  - Reach Lengths
  - Energy Loss Coefficients
  - Stream Junction Data
- Hydraulic structure data (bridges, culverts, etc.), which are also considered geometric data, will be described in later lectures

4

Discussed in this lecture:

The River System Schematic

Cross Section Geometry

Optional Cross Section Properties

Reach Lengths

Energy Loss Coefficients

Stream Junction Data

Hydraulic structure data (bridges, culverts, etc.), which are also considered geometric data, will be described in later lectures

The River System Schematic

WEST CONSULTANTS, INC. www.westconsultants.com

- The river system schematic must be developed before any other data can be entered
- Defines how the river reaches are connected and establishes a naming convention for all the other data
- Developed by drawing and connecting the reaches of the system within the geometric data editor
- You can interact with the schematic by using the viewing options available
  - Includes zooming in/out, displaying river stationing, background pictures, etc.

5

General

The river system schematic is required for any geometric data set and must be developed before any other geometric data can be entered.

The schematic defines how the various river reaches are connected and establishes a naming convention for referencing all the other data.

The river system schematic is developed by drawing and connecting the various reaches of the system within the geometric data editor.

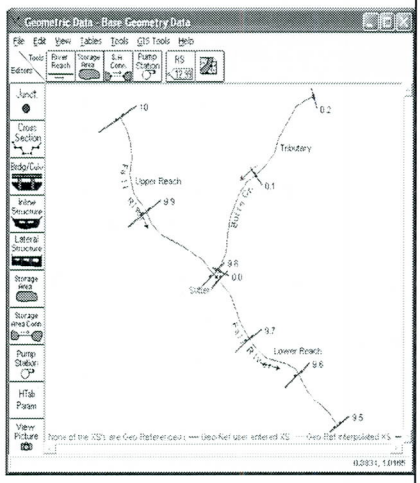
You can interact with the river schematic by zooming in and out, adding background pictures, etc.

The River System Schematic

WEST CONSULTANTS, INC.
www.westconsultants.com

River System and Reach Identifier and Location

- A **river** can be comprised of one or more reaches.
- Each river and reach is given a **unique identifier**
- Other data (e.g., cross sections, bridges) are referenced to a specific reach
  - Each cross section is assigned to a "river" and "reach" identifier:
    River = "Fall River"
    Reach = "Upstream"
    River station = "9.9"
- Reaches are **drawn upstream to downstream** and assumed to be the positive flow direction
- Start or end at open ends of the river system or where two or more reaches join or split
- Reaches **must have at least two points**; the start and the end. Typically drawn with several points that follow the thalweg



6

River System and Reach Identifier and Location

Each river and reach on the schematic is given a unique identifier

Examples:	*River*	*Reach*
	"Fall River"	"Upstream"
	"Fall River"	"Downstream"
	"Butte Cr"	"Butte Cr"

Other data (e.g., cross-sections, bridges) are referenced to a specific reach of the schematic

For example, each cross section is assigned to a "river" and "reach" identifier:

River = "Fall River"
Reach = "Upstream"
River station = "9.9"

Reaches are drawn upstream to downstream which is assumed to be the positive flow direction

A river can be comprised of one or more reaches.

Reaches start or end at open ends of the river system being modeled or at locations where two or more streams join together or split apart

Reaches are drawn as multi-segmented lines and must have at least two points, defining the start and end of the reach.

They are typically drawn with several points that follow the main channel alignment (thalweg).

The River System Schematic

WEST CONSULTANTS, INC. www.westconsultants.com

Adding Tributaries into an Existing Reach

- A tributary is defined as a segment of the river entering or leaving a main stem and contains cross sections
- Locations where water enters or leaves a mainstem, but is not modeled, is defined as local inflows or outflows
- To add a tributary to the middle of an existing reach, draw the new reach and connect it graphically to the existing reach at the location of the new junction
- Enter a River and Reach identifier for the new reach at the prompt
- Verify if you want to "Split" the existing reach into two reaches
  - If "yes", enter a **Reach identifier** for the lower portion of the existing reach and a **Junction name** for the new stream junction

7

Adding Tributaries into an Existing Reach

In these lectures, a tributary (or distributary) is defined as a segment of the river entering or leaving a main stem and contains cross sections

Locations where water enters or leaves a mainstem, but is not modeled, will be defined as local inflows or outflows

To add a tributary (or distributary) to the middle of an existing reach, draw the new reach and connect it graphically to the existing reach at the location of the new junction

The User is then prompted to enter a River and Reach identifier for the new reach

The User then verifies if they want to "Split" the existing reach into two reaches

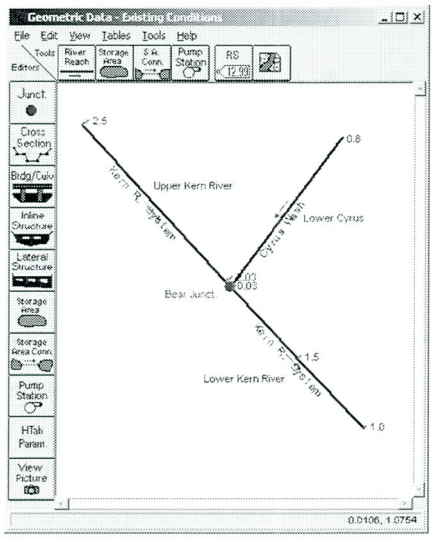
> If "yes", then the User is prompted to enter a Reach identifier for the lower portion of the existing reach and a Junction name for the newly formed stream junction

The River System Schematic

WEST CONSULTANTS, INC.
www.westconsultants.com

Junctions

- The connection of reaches are considered junctions.
- Junctions should not be established with a single reach flowing into another single reach. These two reaches should be combined and defined as one reach.



Junctions

The connection of reaches are considered junctions.

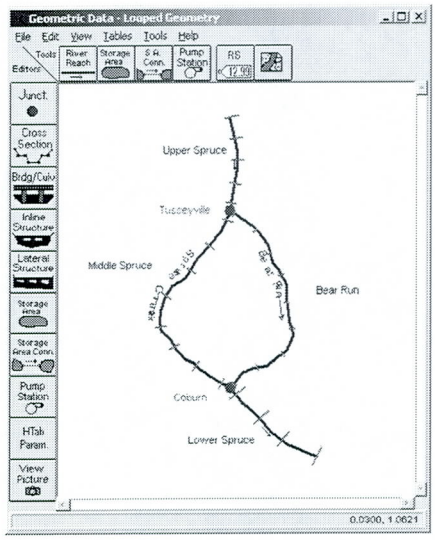
Junctions should not be established with a single reach flowing into another single reach. These two reaches should be combined and defined as one reach.

An example river system schematic

The River System Schematic

WEST CONSULTANTS, INC.
www.westconsultants.com

- Arrows are automatically drawn on the schematic in the assumed positive flow direction
- Junctions are automatically formed as reaches are connected
- A "network" model is where river reaches split apart and then comes back together, forming looped systems



Arrows are automatically drawn on the schematic in the assumed positive flow direction

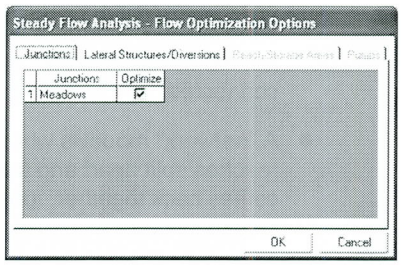
Junctions are automatically formed as reaches are connected

A "network" model is where river reaches split apart and then comes back together, forming looped systems

Optimization of Steady Flow Distributions

WEST CONSULTANTS, INC.
www.westconsultants.com

- HEC-RAS has the capability to optimizing flow splits in steady flow mode at:
  - Lateral structures (levees)
  - Hydraulic connections
  - Storage areas
  - Stream junctions



- In the "Steady Flow Analysis" window, click "Options" on the tool bar menu, and then "Split Flow Optimizations"

10

HEC-RAS has the capability to optimizing flow splits in steady flow mode at:

Lateral structures (levees)

Hydraulic connections

Storage areas

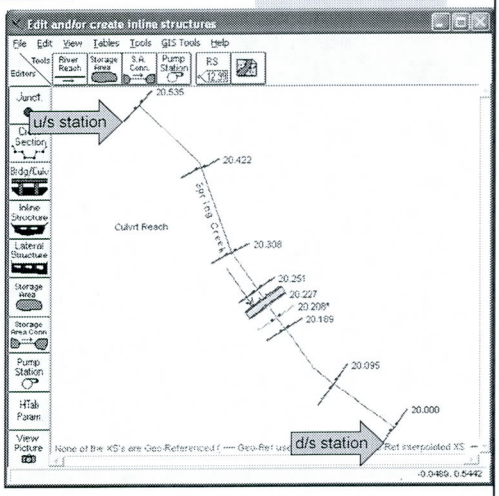
Stream junctions

In the "Steady Flow Analysis" window, click "Options" on the tool bar menu, and then "Split Flow Optimizations"

Ground Points

WEST
CONSULTANTS, INC.
www.westconsultants.com

- Layout and Numbering of Cross Sections
  - Geometry includes cross sections, cross section parameters, and reach lengths.
  - River station identifier is stationing along the channel, in river miles, feet, or any fictitious numbering system.
  - Highest river station is the most upstream section and the lowest is the most downstream section.



11

Ground Points

Layout and Numbering of Cross Sections

Geometry for the analysis of flow in natural streams is specified in terms of ground surface profiles (cross sections) and the measured distances between them (reach lengths).

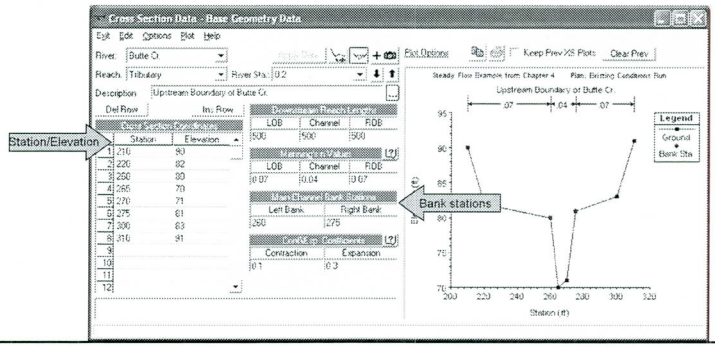
The River Station identifier may correspond to stationing along the channel, in river miles, feet, or any fictitious numbering system.

The numbering system must be consistent; i.e., the program assumes that higher numbers are upstream and lower ones are downstream within a reach.

Ground Points

WEST CONSULTANTS, INC.
www.westconsultants.com

- Data Entry
  - A cross section is described by entering station/elevation (X-Y data) from left to right, with respect to looking downstream.
  - Left side of the stream has the lowest station numbers and the right side has the highest.
  - The left and right bank stations must be specified.
  - Up to 500 data points may be used to in each cross section.
  - Cross section data can have negative stationing values.



12

Data Entry

The cross section is described by entering the station and elevation (X-Y data) from left to right, with respect to looking in the downstream direction.

The program considers the left side of the stream to have the lowest station numbers and the right side to have the highest.

The left and right bank stations separating the main channel from the overbank areas must be specified on the cross section data editor.

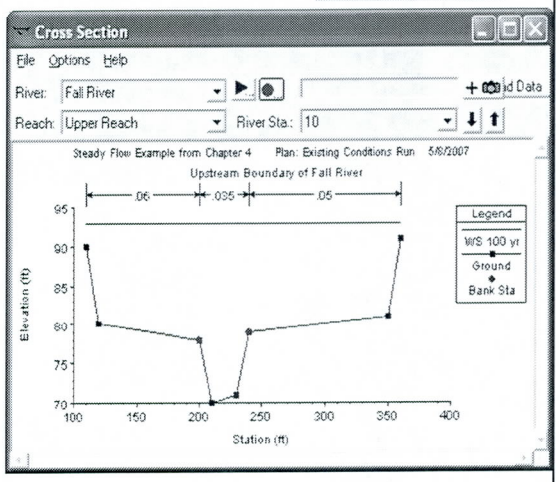
Up to 500 data points may be used to describe each cross section and more than one point can have the same stationing value.

There is a cross section point filter option to reduce the number of points in the X-Y data.

Ground Points

WEST CONSULTANTS, INC. www.westconsultants.com

- Extension of End Points
  - End points of a cross section that are below the computed WSEL are automatically extended vertically.
  - The program adds additional wetted perimeter for any water that comes into contact with the extended vertical walls.



13

Extension of Geometry End Points

End points of a cross section that are too low (below the computed water surface elevation) will automatically be extended vertically.

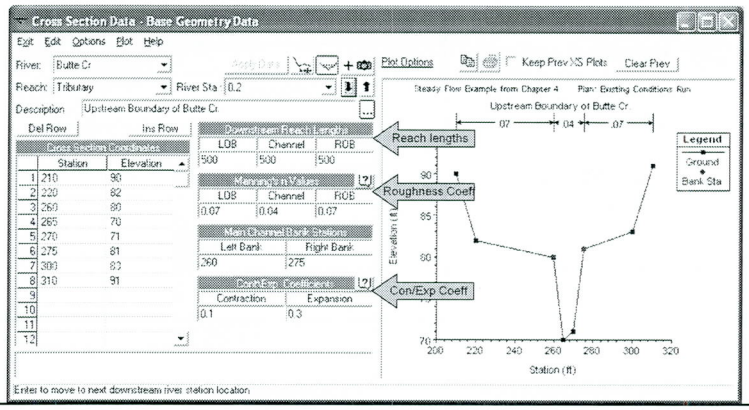
A note indicating that the cross section had to be extended will show up in the output for that section.

The program adds additional wetted perimeter for any water that comes into contact with the extended walls.

Additional Geometric Data

WEST CONSULTANTS, INC.
www.westconsultants.com

- Other data that are required for each cross section consist of:
  - downstream reach lengths;
  - roughness coefficients;
  - contraction and expansion coefficients.



14

Other Geometric Data

Other data that are required for each cross section consist of: downstream reach lengths; roughness coefficients; and contraction and expansion coefficients.

These data will be discussed in detail later.

Reach Lengths

WEST CONSULTANTS, INC. www.westconsultants.com

- The measured distances between cross sections are referred to as reach lengths
  - The reach lengths for the left overbank, right overbank and channel are specified on the cross section data editor
- Channel reach lengths are typically measured along the thalweg
- Overbank reach lengths should be measured along the anticipated path of the center of mass of the overbank flow
- Often, these three lengths will be of similar value.
- There are conditions where they may differ significantly, such as at river bends, or where the channel meanders and the overbanks are straight

15

The measured distances between cross sections are referred to as reach lengths

The reach lengths for the left overbank, right overbank and channel are specified on the cross section data editor

Channel reach lengths are typically measured along the thalweg (deepest part of the channel)

Overbank reach lengths should be measured along the anticipated path of the center of mass of the overbank flow

Often, these three lengths will be of similar value.

There are conditions where they may differ significantly, such as at river bends, or where the channel meanders and the overbanks are straight (DRAW EXAMPLE)

Specify a discharge-weighted reach length based on the discharges in the main channel and left and right overbank segments of the reach

Energy Loss Coefficients

WEST CONSULTANTS, INC. www.westconsultants.com

Several types of loss coefficients are utilized by the program to evaluate energy losses:

- Manning's n values for friction loss
- Contraction and expansion coefficients to evaluate transition (shock) losses
- Bridge and culvert loss coefficients to evaluate losses related to (covered in later lecture):
  - Weir shape
  - Pier configuration
  - Pressure flow
  - Entrance and exit conditions

16

Several types of loss coefficients are utilized by the program to evaluate energy losses:

Manning's n values for friction loss

Contraction and expansion coefficients to evaluate transition (shock) losses

Bridge and culvert loss coefficients to evaluate losses related to:

Weir shape

Pier configuration

Pressure flow

Entrance and exit conditions

Manning's n

WEST CONSULTANTS, INC. www.westconsultants.com

- 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	Channel irregularities	Vegetation
Channel alignment	Scour and deposition	Obstructions
Size and shape of the channel	Stage and discharge	Temperature
Suspended material and bedload	Seasonal change	

- **References** for Manning's n values (some with color photos and calibrated n values for a range of flows)
  - Chow's book: "Open-Channel Hydraulics" [Chow, 1959]
  - Fasken, 1963
  - Barnes, 1987
  - Hicks and Mason, 1991

17

Manning's n

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	Channel irregularities	Vegetation
Channel alignment	Scour and deposition	Obstructions
Size and shape of the channel	Stage and discharge	Temperature
Suspended material and bedload	Seasonal change	

References that show Manning's n values for typical channels (HANDOUT REFERENCES)

Chow's book: "Open-Channel Hydraulics" [Chow, 1959]
Fasken, 1963
Barnes, 1987
Hicks and Mason, 1991

In general, these references provide color photos with tables of calibrated n values for a range of flows

Contraction and Expansion Coefficients

WEST CONSULTANTS, INC.
www.westconsultants.com

- Contraction or expansion of flow due to changes between cross sections is a cause of energy loss
- The loss is computed by applying the C/E coefficients between cross sections. This is defined on the cross section data editor as part of the **upstream cross section.**
  - Small change: Contraction and expansion = 0.1/0.3, respectively
  - Abrupt change: Contraction and expansion = 0.3/0.5, respectively (e.g., bridges)
  - Very abrupt change: Contraction and expansion = 0.6/0.8, respectively (e.g., culverts)
- These values may be changed at any cross section
- Maximum value is 1.0

18

Contraction and Expansion Coefficients

Contraction or expansion of flow due to changes between two cross-sections is a common cause of energy losses within a reach

The loss is computed from the contraction and expansion coefficients specified on the cross section data editor

The coefficients are applied between cross sections and are specified as part of the data for the upstream cross section

Small change: Contraction and expansion = 0.1/0.3, respectively

Abrupt change: Contraction and expansion = 0.3/0.5, respectively

E.g., Bridges

More abrupt change: Contraction and expansion = 0.6/0.8, respectively

E.g., Culverts

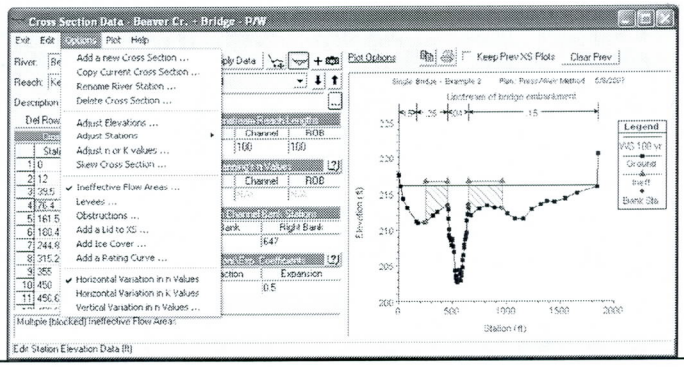
Maximum value for the contraction and expansion coefficient = 1.0

These values may be changed at any cross section

Optional Cross Section Properties

WEST CONSULTANTS, INC.
www.westconsultants.com

- A series of program options are available to restrict flow to the effective flow areas of cross sections. These capabilities are available from the "Options" menu of the Cross Section Data editor.
  - Ineffective Flow areas
  - Levees
  - Blocked Obstructions



19

A series of program options are available to restrict flow to the effective flow areas of cross sections. All of these capabilities are available from the "Options" menu of the Cross Section Data editor.

Ineffective Flow Areas

WEST CONSULTANTS, INC. www.westconsultants.com

- **Ineffective Flow** – Defines areas of the cross section that will contain water but are not being conveyed.
- Typically represents portions of a cross section where water will pond, but the velocity in the downstream direction is close to zero.
- Ineffective flow areas are included in the storage calculations (unsteady flow), but are not included in the active flow area.
- No additional wetted perimeter is added to the active flow area.

20

Ineffective Flow Areas

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.

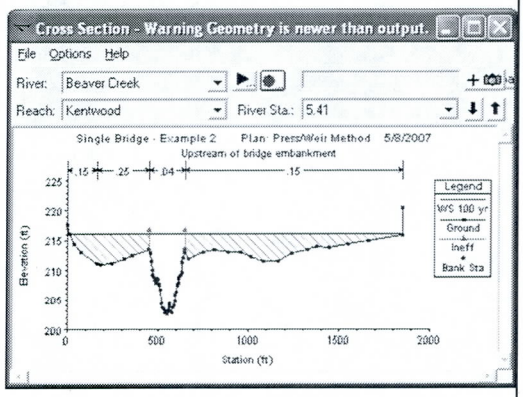
When using ineffective flow areas, no additional wetted perimeter is added to the active flow area.

An example of an ineffective flow area is shown in the next Figure. The cross-hatched area on the left of the plot represents what is considered to be the ineffective flow area.

Ineffective Flow Areas

WEST CONSULTANTS, INC. www.westconsultants.com

- Two types of ineffective flow areas:
  - **Normal Ineffective Areas –** Define a left and/or right station and elevation.
  - **Blocked Ineffective Areas –** Define up to 10 blocked ineffective areas by entering an elevation, left station, and right station for each block.
- If WSEL is below ineffective elevations, areas outside the defined stations are ineffective.
- If WSEL exceeds ineffective elevations, that area is no longer considered ineffective.
- Permanent ineffective flow areas can be set that remain ineffective regardless of WSEL.



21

Two alternatives are available for setting ineffective flow areas.

The first 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 ineffective elevations, the areas to the left of the left station and to the right of the right station are considered ineffective.

Once the water surface goes above either of the established elevations, then that specific area is no longer considered ineffective.

The second option allows for up to 10 **blocked** ineffective flow areas. Blocked ineffective flow areas require the user to enter an elevation, a left station, and a right station for each ineffective block.

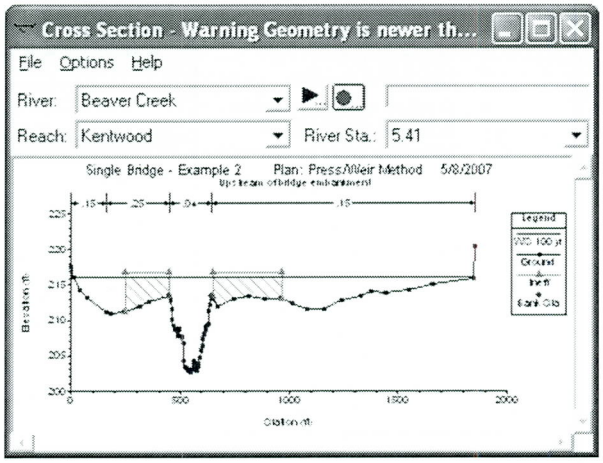
Once the water surface goes above the elevation of the blocked ineffective flow area, the blocked area is no longer considered ineffective.

Permanent ineffective flow areas can be set to remain ineffective regardless of the WSEL.

Ineffective Flow Areas

WEST CONSULTANTS, INC.
www.westconsultants.com

Cross-section with blocked ineffective flow areas



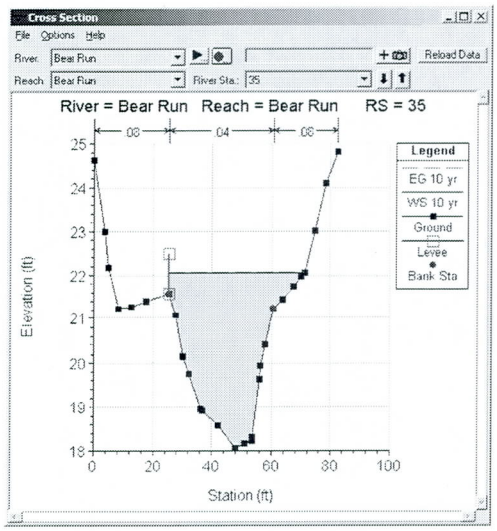
22

Shows blocked ineffective flow areas.

Levees

WEST CONSULTANTS, INC.
www.westconsultants.com

- Define a left and/or right levee station and elevation
- A "vertical wall" is placed at that station up to the defined elevation
- Water cannot go outside the levee station until the levee elevation is exceeded.
- Additional **wetted perimeter is included** when water comes into contact with the levee wall.



Levees

This option allows the user to establish a left and/or right levee station and elevation on any cross section.

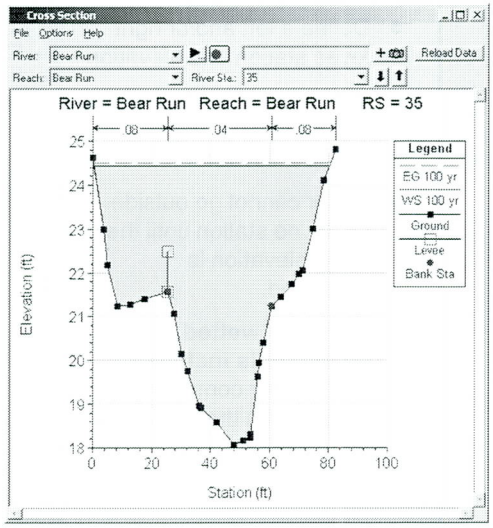
When levees are established, no water can go to the left of the left levee station or to the right of the right levee station until either of the levee elevations are exceeded.

Additional wetted perimeter is included when water comes into contact with the levee wall.

Levees

WEST CONSULTANTS, INC.
www.westconsultants.com

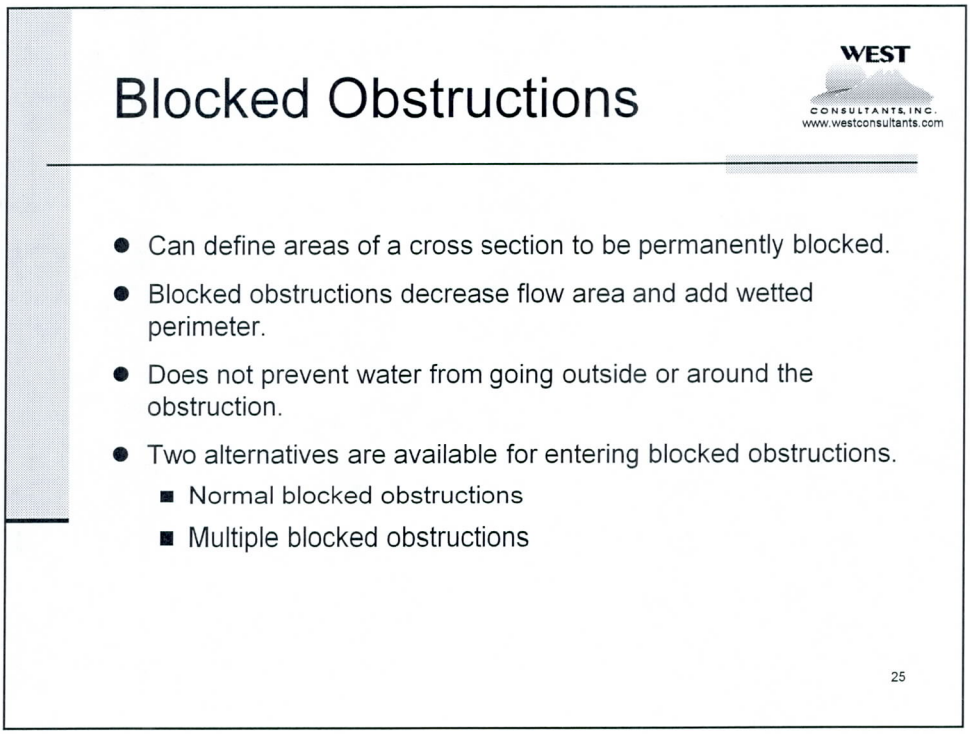
- When levee elevation is exceeded, flow can access the entire cross section
- Wetted perimeter is still added for **BOTH** sides of the levee wall



The user may want to add levees into a data set in order to see what effect a levee will have on the water surface. A simple way to do this is to set a levee station and elevation that is above the existing ground.

If a levee elevation is placed above the existing geometry of the cross section, then a vertical wall is placed at that station up to the established levee height.

Additional wetted perimeter is included for BOTH sides of the levee wall when the levee elevation is exceeded.



Blocked Obstructions

This option allows 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.

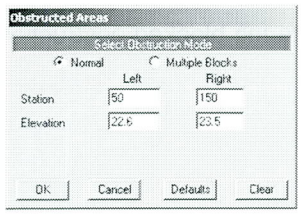
This option does not prevent water from going outside of the obstruction.

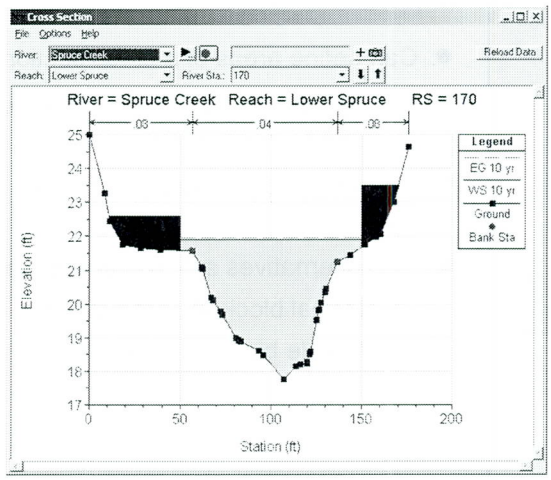
Two alternatives are available for entering blocked obstructions: normal and multiple blocks.

Normal Blocked Obstructions

WEST CONSULTANTS, INC.
www.westconsultants.com

- Define a left and/or right station and elevation
- Area outside the defined stations is completely blocked out





26

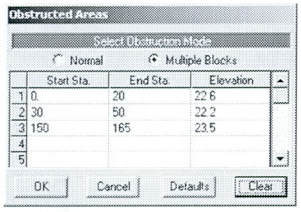
The first option allows the user to define a left station and elevation and a right station and elevation (normal blocked areas).

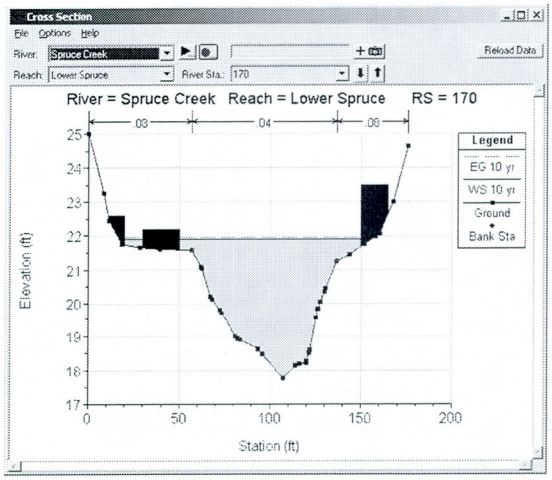
When this option is used, the area to the left of the left station and to the right of the right station will be completely blocked out.

Multiple Blocked Obstructions

WEST CONSULTANTS, INC.
www.westconsultants.com

- Can enter up to 20 individual blocks
- Enter a left station, right station, and an elevation for each block
- Flow is only blocked between the defined stations





27

The second option, for blocked obstructions, allows the user to enter up to 20 individual blocks (Multiple Blocks).

With this option, the user enters a left station, a right station, and an elevation for each of the blocks.

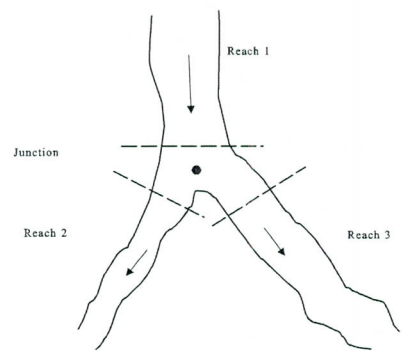
Water can still flow around and between the blocks.

Wetted perimeter is added anywhere the water comes in contact with the blocks.

Stream Junction Data

WEST CONSULTANTS, INC.
www.westconsultants.com

- **General Definition**
  - Stream junctions are locations where two or more streams come together or split apart.
  - Junction data consists of:
    - Reach lengths across the junction
    - Tributary angles (if the momentum method is selected).
  - Cross sections that bound a junction should be placed as close together as possible



28

General Definition

Stream junctions are defined as locations where two or more streams come together or split apart.

Junction data consists of reach lengths across the junction and tributary angles (only if the momentum equation is selected).

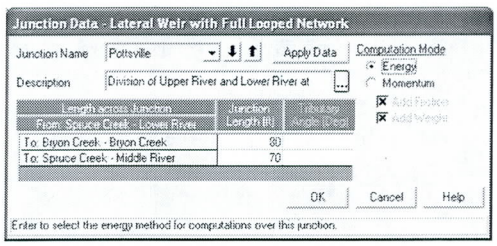
In general, the cross sections that bound a junction should be placed as close together as possible

An example of this is shown in the figure to the right

Stream Junction Data

WEST CONSULTANTS, INC. www.westconsultants.com

- Reach lengths across a junction are entered in the Junction Data editor. This allows for a separate length across the junction for each flow split to be accommodated.
- Two lengths would be entered from the previous figure:
  - Length from most d/s cross section in Reach 1 to most u/s cross section in Reach 2
  - Length from most d/s cross section in Reach 1 to most u/s cross section in Reach 3



29

Reach Lengths Across Junctions

Reach lengths across the junction are entered in the Junction Data editor. This allows for a separate length across the junction for each flow split reach to be accommodated.

Using downstream reach lengths, for the last cross section in Reach 1 from the previous Figure, would not adequately describe the lengths across the junction

It is therefore necessary to describe lengths across junctions in the Junction Data editor

In the previous example, two lengths would be entered

> These lengths should represent the average distance that the water will travel from the most downstream cross section in Reach 1 to the most upstream cross section of Reach 2 and Reach 3

Junction Modeling Methods

WEST CONSULTANTS, INC. www.westconsultants.com

- A junction can be modeled by the energy equation or the momentum equation.
  - ***Energy Method***
    - The energy equation does not take into account the angle of any tributary entering or exiting the main stream.
    - In most cases, the energy loss due to the tributary flow angle is insignificant, and using the energy method is adequate.
  - ***Momentum Method***
    - There are situations where the angle of the tributary can cause significant energy losses and the momentum method is a more appropriate selection.
    - When the momentum method is selected, an angle for all tributaries of the main stem must be entered.
    - A detailed description of junction calculations can be found in Chapter 4 of the Hydraulic Reference Manual.

30

In HEC-RAS, a junction can be modeled by either the energy equation or the momentum equation.

Energy Method

The energy equation does not take into account the angle of any tributary coming in or leaving the main stream, while the momentum equation does.

In most cases, the energy loss due to the tributary flow angle is insignificant, and using the energy equation to model the junction is more than adequate.

Momentum Method

There are situations where the angle of the tributary can cause significant energy losses. In these situations, it would be more appropriate to use the momentum approach.

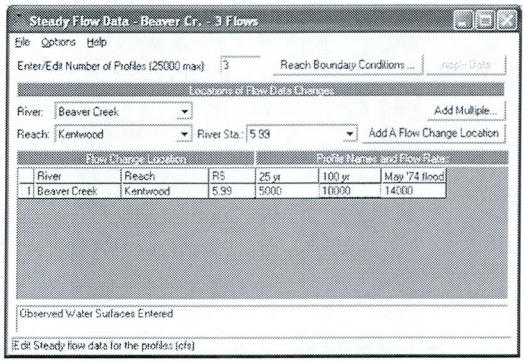
When the momentum approach is selected, an angle for all tributaries of the main stem must be entered.

A detailed description of how junction calculations are made can be found in Chapter 4 of the Hydraulic Reference Manual.

Steady Flow Data

WEST CONSULTANTS, INC.
www.westconsultants.com

- Steady flow data are required in order to perform a steady water surface profile calculation and consist of:
  - Flow Regime
  - Boundary Conditions
  - Discharge Information



31

Steady flow data are required in order to perform a steady water surface profile calculation. Steady flow data consist of:

Flow Regime

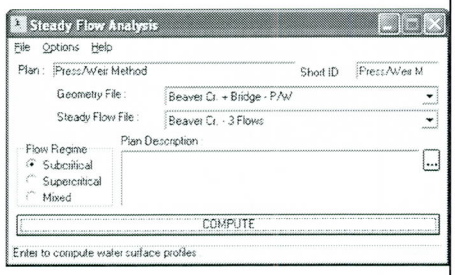
Boundary Conditions

Discharge Information

Flow Regime

WEST CONSULTANTS, INC.
www.westconsultants.com

- Flow regime (subcritical, supercritical, or mixed) is specified on the Steady Flow Analysis window
- Profile computations begin at the boundary conditions
- **Subcritical flow**
  - WSEL ≥ critical depth
  - calculations begin at d/s end
- **Supercritical flow**
  - WSEL ≤ critical depth
  - calculations begin at u/s end
- **Mixed flow**
  - WSEL is not constrained
  - calculations begin at both d/s and u/s ends



32

The flow regime (subcritical, supercritical, or mixed flow regime) is specified on the Steady Flow Analysis window of the user interface

Profile computations begin at a cross section with known or assumed starting conditions and proceed upstream for subcritical flow or downstream for supercritical flow

Subcritical profiles computed by the program are constrained to critical depth or above, and supercritical profiles are constrained to critical depth or below

In cases where the flow regime will pass from subcritical to supercritical, or vice versa, the program should be run in a mixed flow regime mode

Boundary Conditions

WEST CONSULTANTS, INC. www.westconsultants.com

- **General**
  - Necessary to establish the starting water surface at the ends of the river system (upstream and downstream) to begin calculations
- **Boundary Condition Requirements by Flow Regime**
  - Subcritical flow – downstream boundary conditions required
  - Supercritical flow – upstream boundary conditions required
  - Mixed flow – upstream and downstream boundary conditions required

33

General

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

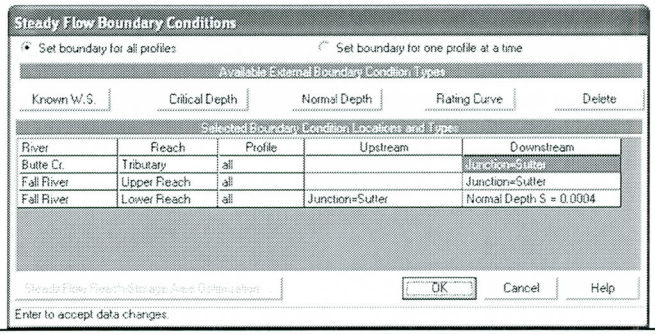
If a supercritical flow regime is going to be calculated, boundary conditions are only necessary at the *upstream ends* of the river system

If a mixed flow regime calculation is going to be made, then boundary conditions must be entered at *all ends* of the river system

Entering Boundary Condition Data

WEST CONSULTANTS, INC.
www.westconsultants.com

- The boundary conditions editor lists each reach with an upstream and a downstream boundary condition.
- Junctions are internal boundary conditions and are automatically listed in the table.
- The user is required to enter the necessary external boundary conditions.



34

Entering Boundary Condition Data

The boundary conditions editor contains a table listing every reach.

Each reach has an upstream and a downstream boundary condition.

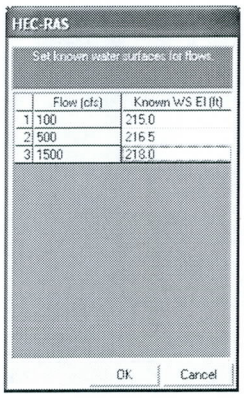
Connections to junctions are considered internal boundary conditions. Internal boundary conditions are automatically listed in the table, based on how the river system was defined in the geometric data editor.

The user is required to enter the necessary external boundary conditions. There are four types of boundary conditions available to the user:

Boundary Condition Types

WEST CONSULTANTS, INC. www.westconsultants.com

- Four types of boundary conditions are available:
  - **Known Water Surface Elevations**
    - ▶ User must enter a known water surface elevation for each profile
  - **Critical Depth**
    - ▶ User is not required to enter any further information. The program will calculate critical depth for each profile and use that as the boundary condition



35

1. Known Water Surface Elevations

For this boundary condition, the user must enter a known water surface elevation for each of the profiles to be computed.

2. Critical Depth

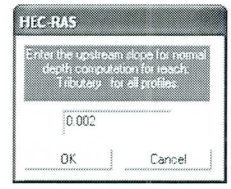
When this type of boundary condition is selected, the user is not required to enter any further information

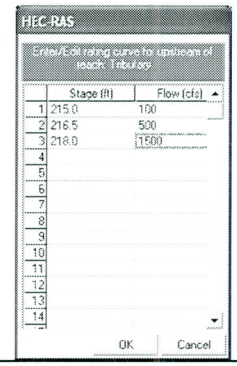
The program will calculate critical depth for each of the profiles and use that as the boundary condition

Boundary Condition Types (cont'd)

WEST CONSULTANTS, INC. www.westconsultants.com

- **Normal Depth**
  - ▶ User is required to enter an energy slope to be used to calculate normal depth for each profile
  - ▶ In general, the energy slope can be approximated by using the average bed slope or the average water surface slope near the boundary condition
- **Rating Curve**
  - ▶ User is required to enter an elevation versus flow rating curve and the starting elevation is interpolated from the rating curve for each profile





36

3. Normal Depth

For this type of boundary condition, the user is required to enter an energy slope that will be used in calculating normal depth (Manning's equation) at that location

A normal depth will be calculated for each profile based on the user-entered slope

In general, the energy slope can be approximated by using the average slope of the channel or the average slope of the water surface in the vicinity of the cross section

4. Rating Curve

When this type of boundary condition is selected, a popup window appears allowing the user to enter an elevation versus flow rating curve

For each profile, the starting elevation is interpolated from the rating curve

Guidelines for Estimating Starting WSEL

WEST CONSULTANTS, INC. www.westconsultants.com

- Using an estimated water surface can incorporate an error in the water surface profile in the vicinity of the boundary condition.
- If it is important to have accurate answers at cross sections near the boundary condition, additional cross sections should be added.
- ***Locations of Additional Cross Sections by Flow Regime***
  - Subcritical – add cross sections downstream of the study reach.
  - Supercritical – add cross sections upstream of the study reach.
  - Mixed flow – add cross sections upstream and downstream of the study reach.

37

Guidelines for Estimating Starting WSEL

Using an estimated water surface can often incorporate an error in the water surface profile in the vicinity of the boundary condition.

If it is important to have accurate answers at cross sections near the boundary condition, additional cross sections should be added.

Locations of Additional Cross Sections by Flow Regime

only	If a subcritical profile is being computed, then additional cross sections need be added below the downstream boundaries.
should	If a supercritical profile is being computed, then additional cross sections be added upstream of the relevant upstream boundaries.
be	If a mixed flow regime profile is being computed, then cross sections should added upstream and downstream of all the relevant boundaries.

Boundary Conditions

WEST CONSULTANTS, INC. www.westconsultants.com

- ***Check for Sufficiency of Additional Cross Sections***
  - A sensitivity test should be run to test whether the added cross sections are sufficient for a particular boundary condition.
  - Try several different boundary conditions for a single profile.
  - If the water surface profile converges by the time the computations reach the study area cross sections, the boundary condition is not affecting answers in the study area.
  - **Initial suggestion**: Add four cross sections, each spaced approximately five active channel widths apart.

38

Check for Sufficiency of Additional Cross Sections

In order to test whether the added cross sections are sufficient for a particular boundary condition, the User should try several different starting elevations at the boundary condition, for the same discharge.

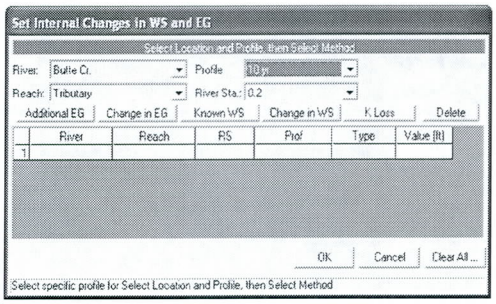
If the water surface profile converges to the same answer, by the time the computations get to the cross sections that are in the study area, then enough sections have been added, and the boundary condition is not affecting the answers in the study area.

A suggested initial number of cross sections is four and spaced approximately 5 active channel widths (distance between left and right bank stations) apart.

Other Options of the Steady Flow Data Menu

WEST CONSULTANTS, INC.
www.westconsultants.com

- Multiply selected discharge values by a user specified amount to increase or decrease the values one at a time or globally
- Set specific changes in the water surface and energy between any two cross sections in the model:
  - The changes in water surface and energy can be set for a specific profile in a multiple profile model
  - Four options can be selected:
    - ▶ Additional EG
    - ▶ Change in EG
    - ▶ Known WS
    - ▶ Change in WS



Other Options of the Steady Flow Data Menu

Multiply selected discharge values by a user specified amount to increase or decrease the values one at a time or globally

Set specific changes in the water surface and energy between any two cross sections in the model:

The changes in water surface and energy can be set for a specific profile in a multiple profile model

Four options can be selected: Additional EG, Change in EG, Known WS, and Change in WS

Other Options of the Steady Flow Data Menu (cont'd)

WEST CONSULTANTS, INC. www.westconsultants.com

- Additional EG option allows the User to add an additional energy loss between two cross sections
  - The program will use the specified energy loss in the energy balance equation in addition to the normal losses
- Change in EG option allows the User to set a specific amount of energy loss between two cross sections
  - The program adds the specified energy loss to the energy of the downstream section and computes a corresponding water surface.
- Known WS option allows the User to set a water surface at a specific cross section for a specific profile
  - The program will use the known water surface elevation for any cross section where a known WSEL has been entered
- Change in WS option allows the User to force a specific change in the water surface elevation between two cross sections
  - The program adds the specified change in water surface to the downstream cross section and computes a corresponding energy

40

The Additional EG option allows the User to add additional energy loss between two cross sections

This energy loss will be used in the energy balance equation in addition to the normal friction and contraction and expansion losses

Change in EG option allows the User to set a specific amount of energy loss between two cross sections

When this option is used, the program does not perform an energy balance, it adds the specified energy loss to the energy of the downstream section and computes a corresponding water surface.

Known WS option allows the User to set a water surface at a specific cross section for a specific profile

During the computations, the program will not compute a water surface elevation for any cross section where a known water surface elevation has been entered, but instead will use the known water surface elevation and then move to the next section

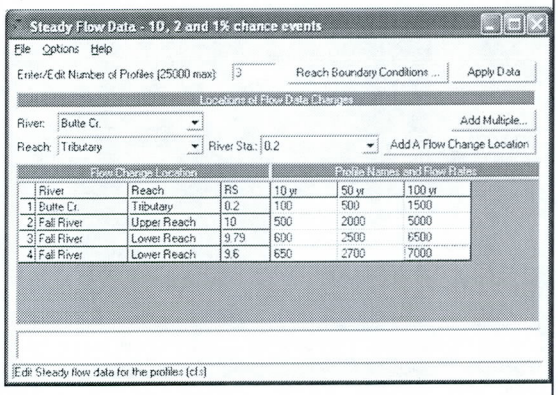
Change in WS option allows the User to force a specific change in the water surface elevation between two cross sections

When this option is selected, the program adds the user-specified change in water surface to the downstream cross section, and then calculates a corresponding energy to match the new water surface

Discharge Information

WEST CONSULTANTS, INC.
www.westconsultants.com

- Flow data are entered from upstream to downstream for each reach
- Flow remains constant until another flow value is entered within the same reach
- The flow rate can be changed at any cross section
- The flow rate cannot be changed in the middle of a bridge, culvert, or stream junction
- Flow data must be entered for all profiles



41

Discharge information is required at each cross section in order to compute the water surface profile

Discharge data are entered from upstream to downstream for each reach

At least one flow value must be entered for each reach in the river system

Once a flow value is entered at the upstream end of a reach, it is assumed that the flow remains constant until another flow value is encountered within the same reach

The flow rate can be changed at any cross section within a reach

The flow rate cannot be changed in the middle of a bridge, culvert, or stream junction

Flow data must be entered for the total number of profiles that are to be computed

WORKSHOP SOLUTIONS

WEST CONSULTANTS, INC.

HEC-RAS WORKSHOP NO. 1

DEVELOPING A HYDRAULIC MODEL

Question 1

What is the default method used for calculating Friction Slope?

Average Conveyance [Steady Flow Analysis window: Options menu, Friction Slope Method]

Question 2

What is the Energy Slope at R.K. 2.0 for the 5-year profile?

0.000660 m/m [Cross Section or Profile Output Table]

Question 3

What is the Top Width at R.K. 1.5 for the 1-year profile? *25.66 m [Cross Section or Profile Output Table]*

Question 4

What are the Total, Left OB (Overbank), Channel, and Right OB Average Velocities (m/s) at R.K. 2.0 for the 5-year profile?

Total: *0.39 m/s* Left OB: *0.36 m/s* Channel: *0.59 m/s* Right OB: *0.21 m/s*

[Cross Section Output Table]

Question 5

What warning is given at R.K. 2.0 for the 5-year profile? *Divided flow [Cross Section Output Table or Summary of Errors, Warnings, and Notes]*

What would you do to correct this, if anything? *Divided flow would be OK if the water is divided by a high ground level, which looks correct given the geometry. However, a field inspection and engineering judgment is required to determine whether the water can really flow in the overbanks under all profiles. Options are available in the Cross Section Data Editor to control what areas of the cross section are available for flow.*

Question 6

What is the Flow Area at R.K. 1.0 for the 10-year profile? *29.79 m^2 [Profile Output Table]*

This document is property of WEST Consultants, Inc., and is not to be duplicated without prior consent.

Question 7

What is the Water Surface Elevation (m) at R.K. 2.0 for each profile?

1-year: *3.96 m* 5-year: *4.84 m* 10-year: *5.48 m*

[Profile Output Table]

Question 8

How close do the cross sections need to be to eliminate the warnings for more cross sections? *A cross section every 100 meters eliminates the warnings for more cross sections.*

This document is property of WEST Consultants, Inc., and is not to be duplicated without prior consent.

WEST CONSULTANTS, INC.

HEC-RAS Workshop No. 1

Developing a Hydraulic Model

Objective: Learn how to create a new project, enter geometric, hydraulic, and flow data, create a new plan, run a steady flow analysis, and view results.

Time to Complete: 50 to 60 minutes

Preface

HEC-RAS Workshops

Welcome to the first HEC-RAS workshop! We have tried to make these workshops as easy-to-follow and informative as possible. Each workshop is divided into a number of major steps, with detailed instructions provided for each step. In addition, the instructions are interspersed with Notes and Warnings, which provide additional guidance on completing the workshop or on using the HEC-RAS program in general. The symbols designating Notes and Warnings are shown below.

*The **Note** symbol represents useful additional information, such as an alternate way to perform a function in HEC-RAS.*

*The **Warning** symbol represents important or critical information, such as avoiding a potential fatal error in HEC-RAS.*

Geometric Data Editor

Also, note that at first when the instructions say to click an icon on the toolbar, this icon is shown graphically in the left margin, like the example shown to the left.

Finally, you will be able to test your new HEC-RAS skills by answering questions near the end of the workshop. Questions are designated by the symbol shown below.

Question 1

*The **Question** symbol represents an opportunity to test your HEC-RAS knowledge.*

After each workshop, your instructor will go over the workshop solutions and answer any additional questions that you might have.

Intro

Steps for Workshop No. 1

The first HEC-RAS workshop – Developing a Hydraulic Model – is divided into seven steps:

Step 1. Create a project directory and create a new project

Step 2. Create a new river reach

Step 3. Create a new cross section

Step 4. Create additional cross sections

Step 5. Enter flow data and reach boundary conditions

Step 6. Create a plan

Step 7. Run the steady flow analysis and view results

Step 1

Create a Project Directory and Create a New Project

If you haven't already done so, install the workshop files from the CD-ROM to the C:\ drive, and start the HEC-RAS program. Now you're ready to begin Workshop No. 1!

(a) From the File menu, select **New Project**.

(b) Browse for and then double-click on **C:\Workshops\Basic_RAS_Workshops** in the directory window. Click the **Create Folder** button. Type **Workshop_1** for the name of the project directory.

(c) Type **Developing a Hydraulic Model** as the title.

(d) Enter **WK1_Develop.prj** as the project file name, and then click **OK**.

*If you get the message **Title and file name must be specified**, you may have an asterisk (*) in the file name. Delete the asterisk, and then click **OK** again.*

A message window will appear asking for confirmation of the file name, title, directory, and units system.

(e) Click **OK** at the confirmation message.

You will be returned to the HEC-RAS main menu screen.

(f) Enter a suitable description (e.g., **Workshop No. 1 - Developing a Hydraulic Model**) in the Project Description window.

(g) From the Options menu, select **Unit system (U.S. Customary/SI)**.

(h) Select **System International (Metric System)**, select **Set as default for new projects**, and then click **OK**.

A message window will appear asking you to confirm that you want to set the project units to SI (metric). Click **Yes**.

Save Project

(i) Click the **Save Project** icon on the toolbar.

Step 2 Create a New River Reach

The next step in developing a hydraulic model is to create a geometric data file.

(a) From the Edit menu, select **Geometric Data**.

Geometric Data Editor

You can also access the Geometric Data editor by clicking the geometric data icon on the toolbar.

This will bring up the Geometric Data window.

(b) Click the **River Reach** icon (located on the top of the screen). Move the cursor – which will now look like a pencil – to the top left corner of the window.

(c) Click the left mouse button once and move the pencil to the bottom right corner of the window.

When creating a new river reach, each single click of the mouse creates a new point on the reach. The points are for the schematic figure only and do not necessarily correspond to any cross section locations.

(d) Double click the left mouse button to complete the reach.

This will bring up a window in which to name the river and reach.

(e) Enter the name of the River as **Kern R. System** and the reach as **Lower Kern River**.

*If you accidentally clicked the mouse too many times and created extra points along the reach, you can remove them by selecting **Remove Points in a Reach or SA** from the Edit menu (a minus symbol and circle is added to the mouse arrow). Just click on the extra point to remove it. When finished, turn off the point removal option by selecting*

Remove Points in a Reach or SA *a second time.* ***SA*** *refers to Storage Areas, usually used in unsteady flow modeling.*

(f) From the File menu on the Geometric Data editor, select **Save Geometry Data**.

(g) Enter **Existing Conditions** for the title and click **OK**.

The file name for the Geometry Data file is selected automatically by HEC-RAS based on the project name.

Step 3 Create a New Cross Section

Now that the river and reach have been created, we are going to create three cross sections. The first will be located at River Kilometer (R.K.) 1.0, the next will be at R.K. 1.5, and the last will be at R.K. 2.0. The cross section at R.K. 2.0 will be created by copying and modifying the cross section at R.K. 1.5.

(a) Click the **Cross Section** button on the left side of the window, which opens the Cross Section Data editor.

(b) From the Options menu, select **Add a new Cross Section**. For the River Station, enter **1.0** and click **OK**.

(c) In the Description box, enter **Located 1.0 kilometer above the confluence with the San Joaquin River**.

View Description

(d) To see the entire description, click the button with the three dots located just to the right of the description window.

(e) Click on the button again to close the expanded window.

(f) Enter the following **Cross-Section X-Y Coordinates** (Station-Elevation data in meters) in the proper fields:

Stations must be entered in an increasing direction (i.e., the station for point 2 must be greater than - or equal to - the station at point 1). The convention for cross section orientation is from left to right, looking downstream.

Use the Return key to enter data in x-y pairs (i.e., station 1, elevation 1, station 2, elevation 2, etc.). Use the up/down arrows on your keyboard to enter data in columns (i.e., station 1, station 2, etc., then elevation 1, elevation 2, etc.).

	Station	Elevation
1	0	5.5
2	15	4.3
3	37	3.7
4	40	4.0
5	41	4.6
6	43	1.8
7	44	1.2
8	46	2.1
9	47	4.6
10	52	6.1

(g) Enter the following **Downstream Reach Lengths (m)**:

LOB	Channel	ROB
0	0	0

The downstream reach lengths are equal to zero because this cross section is located at the downstream end of the reach. There are no cross sections downstream of this one.

From field inspection and other evaluations, we have determined appropriate Manning's n values, as well as the location of the main channel bank stations.

(h) Enter the following **Manning's n Values**:

LOB	Channel	ROB
0.08	0.04	0.06

(i) Enter the following **Main Channel Bank Stations (m)**:

Left Bank	Right Bank
41	47

Leave the default values for the contraction and expansion coefficients.

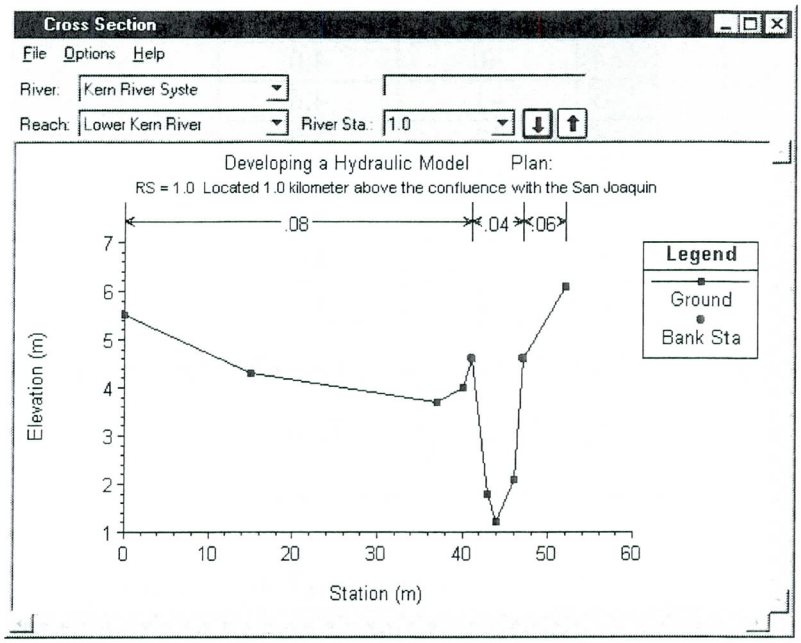
(j) Click **Apply Data**.

Clicking Apply Data puts the values in program memory and updates any graphs or figures. Notice that the text color changes from red to black when data is applied.

Once the cross section data has been entered, it is always a good idea to plot the cross section and verify the data input.

(k) From the Plot menu, select **Plot Cross Section**.

Your cross section plot should look similar to the following:



(l) Close the cross section plot, and then close the Cross Section Data editor.

(m) From the File menu on the Geometric Data editor, select **Save Geometry Data**. This saves the geometric data to the computer hard disk.

Step 4 Create Additional Cross Sections

We now want to add a second cross section, this time at R.K. 1.5.

(a) Open the Cross Section Data editor by clicking the **Cross Section Data** icon on the left side of the screen.

(b) From the Options menu, select **Add a New Cross Section**.

(c) Enter **1.5** for the new river station, then click **OK** or hit **Enter**.

(d) In the Description box, enter **River Kilometer 1.5**.

(e) Enter the following **Cross-Section X-Y Coordinates** (Station-Elevation data in meters) in the proper fields:

	Station	Elevation
1	1	6.7
2	18	3.4
3	40	3.0
4	43	4.6
5	44	2.1
6	45	1.8
7	46	3.0
8	47	5.5
9	49	3.7
10	54	6.4

(f) Enter the following **Downstream Reach Lengths (m)**:

LOB	Channel	ROB
450	500	650

(g) After deleting the default values, enter the following **Manning's n Values**:

LOB	Channel	ROB
0.085	0.035	0.055

From field inspection and other evaluations, we have determined the location of the channel bank stations.

(h) Enter the following **Main Channel Bank Stations (m)**.

Left Bank	Right Bank
43	47

(i) Click **Apply Data**.

(j) From the Plot menu, select **Plot Cross Section**, and verify that plot looks OK. When finished, close the cross section plot and the cross section data editor.

(k) From the File menu on the Geometric Data editor, select **Save Geometry Data**.

We will now add a cross section at River Kilometer 2.0, which is a copy of R.K. 1.5, but is raised by 0.5 meters, with a 10% larger area on the left overbank and a 5% smaller area on the right overbank. The channel area will remain unchanged.

(l) Open the Cross Section Data Editor and confirm that you are at River Station 1.5. If not, scroll through the cross sections using the arrow buttons next to **River Sta**.

(m) From the Options menu, select **Copy Current Cross Section**.

(n) Enter **2.0** for the new river station, and then click **OK**.

(o) Change the description to **River Kilometer 2.0**.

(p) Enter the following **Downstream Reach Lengths (m)**:

LOB	Channel	ROB
400	500	700

(q) Enter the following **Manning's n Values**:

LOB	Channel	ROB
0.07	0.04	0.05

(r) From the Options menu, select **Adjust Elevations**. Enter **0.5**, and then click **OK**.

Note that all of the elevations in the Cross Section X-Y Coordinates table have been adjusted to reflect this change.

(s) From the Options menu, select **Adjust Stations** and **Multiply by a Factor**.

(t) Enter **1.1** (10% increase) for the LOB, **1.0** (no increase) for the Channel, and **0.95** (5% decrease) for ROB. When finished, click **OK** or press **Enter**.

Note that the stations have been changed in the Cross Section X-Y Coordinates table.

(u) Click **Apply Data** on the Cross Section Data Editor.

(v) From the Plot menu, select **Plot Cross Section**.

(w) Click the up or down arrows to the right of the River Sta. heading in the plot screen. Check the relative changes in cross section shape and other features by scrolling through the plots

The scale for the plot axis may change for each cross section. To use the same scale for all cross sections, select ***Set Persistent Scale*** *from the* Options *menu. You can accept the scale definition for any particular cross section by clicking OK, or edit the scale as you wish to capture more cross section data within the same scale limits (i.e. change the y axis minimum to zero in this case.)*

(x) Close the cross section plot, and then close the Cross Section Data editor.

(y) From the File menu on the Geometric Data editor, select **Save Geometry Data**.

Finally, we want to check the invert of the reach.

(z) Click the Cross Section button to open the Cross Section Data editor. From the Plot menu, select **Plot Profile**.

(aa) Close the Profile Plot, Cross Section Data editor, and Geometric Data editor, leaving the main HEC-RAS window.

Step 5 Enter Flow Data and Reach Boundary Conditions

We want to run three subcritical flow profiles (1-year, 5-year, and 10-year discharges). The 1-year profile has a known water surface for the downstream reach boundary condition. For the 5-year and 10-year profiles, normal depth is used as the boundary condition.

(a) From the Edit menu, select **Steady Flow Data**.

Steady Flow Data Editor

You can also access the Steady Flow Data editor by clicking the steady flow data icon on the toolbar.

(b) Enter **3** in the **Enter/Edit Number of Profiles (25,000 max)** box and press **Enter**.

(c) From the Options menu, select **Edit Profile Names**. Delete the default profile names and enter the following names for each of the three profiles. When finished, click **OK**:

	Profile #	Profile Name
1	1	1-year
2	2	5-year
3	3	10-year

(d) Enter the following flow rates (m^3/s) for the three profiles at River Station 2.0 on the Lower Kern River:

1-year	5-year	10-year
3	15	30

(e) Click the **Reach Boundary Conditions** button.

(f) Select **Set boundary for one profile at a time at** the top of the window.

This will add rows for all three profiles. Because we are modeling a subcritical profile, we need to add boundary condition data in the Downstream column only.

(g) Click on the first cell below the **Downstream** column heading (the cell should be highlighted in blue).

(h) Click the **Known W.S.** button, and then enter **2.7** for the downstream known water surface elevation (profile: 1-year). Click **OK**.

(i) Click on the second cell below the **Downstream** column heading (the boundary condition for the 5-year profile).

(j) Click the **Normal Depth** button, and then enter **0.003** for the downstream slope for the normal depth computation (profile: 5-year). Click **OK**.

(k) Click on the third cell below the **Downstream** column heading (the boundary condition for the 10-year profile).

(l) Click the **Normal Depth** button, and then enter **0.003** for the downstream slope for the normal depth computation (profile: 10-year). Click **OK**. Click **OK** again to close the Steady Flow Boundary Conditions window.

(m) Click **Apply Data** and from the File menu, select **Save Flow Data**.

(n) Enter **1, 5, 10 year Qs - Existing Conditions** for the title, then click **OK**.

(o) Close the Steady Flow Data window and return to the main HEC-RAS window.

Step 6 Create a Plan

(a) Click the **Steady Flow Analysis** icon on the toolbar.

Steady Flow Analysis

*This icon is also known as the "running stick man". You can also access the Steady Flow Analysis editor by selecting **Steady Flow Analysis** from the Run menu.*

The last saved geometry and steady flow data files will be listed. In this project, we have created only one geometry file and one steady flow file.

(b) From the File menu, select **New Plan**.

(c) Enter **Existing, 1, 5, and 10 year Qs** for the title, then click **OK**.

(d) Enter **Existing** for the short plan identifier (short ID) and click **OK**.

(e) For the flow regime, click **Subcritical** (it may already be selected).

(f) From the Options menu, select **Critical Depth Output Option**, and then select **Critical Always Calculated**. Click **OK**.

(g) From the File menu, select **Save Plan**.

Question 1 What is the **default method** used for calculating **Friction Slope**?

_____

Hint: Look in the Options menu of the Steady Flow Analysis window.

Step 7 Run the Steady Flow Analysis and View Results

(a) Go to the main HEC-RAS File menu, and **Save Project.**

(b) Return to the Steady Flow Analysis window, and click the large **Compute** button at the bottom.

A window showing the progress of the computations will be displayed.

SNET is the steady flow computation engine used by the HEC-RAS program.

(c) When computations are finished, click the **Close** button on the progress window.

(d) Close the Steady Flow Analysis window to return to the main HEC-RAS window.

Now we're ready to look at output from the run. Graphical, tabular, and text output can be shown by either selecting an option from the View menu, or by clicking the appropriate icons on the toolbar. We'll start by looking at the cross section plots.

Cross Section

(e) Click the **Cross Sections** icon on the toolbar to open the cross section graphical output window.

(f) From the Options menu, select **Profiles**. Click the **Select All** button to show all profiles in the plot, and then click **OK** to confirm.

*If you want to view particular profiles, click the **Clear All** button, and then double-click on the desired profiles in the Avail Profiles box (on the left). This will enter the profiles in the Selected box (on the right). Click **OK** to confirm.*

(g) Scroll through the cross sections using the arrow buttons next to the River Sta. Look at the water surface elevations for each cross section. Note the distribution of flow in the left overbank, channel, and right overbank. We have not restricted the flow from using any portion of each cross section. When you are finished, close the Cross Section window.

Next, we'll look at the water surface profile plot.

W.S. Profile Plot

(h) Click the **Water Surface Profiles** icon on the toolbar to open the profile plot window.

(i) If not all three profiles are shown in the plot; select **Profiles** from the Options menu. Click the **Select All** button to show all profiles in the plot, and then click **OK** to confirm.

(j) If critical depth is not already shown on the plot, select **Variables** from the Options menu. Click on **Critical Depth Elevation** to select it. Take at look at the other variables available for plotting, and then click **OK**. Note that each cross section has a different critical depth for each profile. The critical depth for cross sections 1.5 and 2.0 are affected by flow in the overbanks.

(k) From the Options menu, select **Landmarks** and then **River Stationing**.

(l) When you are finished looking at the profile plot, close the plot window and return to the main HEC-RAS window.

In addition to the cross section and profile plots, there are also general profile plots (velocity, flow area, etc. versus channel distance), rating curves, and XYZ perspective plots.

There are two types of tabular output: cross section and profile tables. First, we'll look at the cross section table.

Cross Section Table

(m) Click the **Cross Section Table** icon on the toolbar.

(n) Using the drop-down arrow next to **Profile** box, select the **5-year** profile.

(o) Using the drop-down arrow (or up/down arrows) next to **River Sta** box, select the **2.0** river station, and then answer Question 2 below.

Question 2 What is the **Energy Slope** at **R.K. 2.0** for the **5-year** profile? _____

Select the appropriate cross sections and profiles to answer Questions 3, 4, and 5.

Question 3 What is the **Top Width** at **R.K. 1.5** for the **1-year** profile? _____

Question 4 What are the **Average Velocities** at **R.K. 2.0** for the **5-year** profile for Total, Left OB (Overbank), Channel, and Right OB?

Total: _____ Left OB: _____ Channel: _____ Right OB: _____

The bottom portion of the Cross Section Table lists any applicable errors, warnings, and notes for the cross section.

Question 5 What warning is given at **R.K. 2.0** for the **5-year** profile? _____

_____

What would you do to correct this, if anything? _____

_____

The second type of tabular output is the profile table. While most of the information is also available in the cross sections table, the profile table allows data from multiple river stations to be viewed at the same time.

Profile Table

(p) Click the **Profile Table** icon on the toolbar.

(q) From the Options, select **Include Profile Name in Table**.

(r) To add more profiles to the table select Options, then **Profiles,** and choose the desired profiles.

If you have multiple profiles, you should always include the profile name in the table. If the project has multiple plans or reaches, these can also be selected from the Options menu of the Profile Table. Now use the profile table to answer Questions 6 and 7.

Question 6

What is the **Flow Area** at **R.K. 1.0** for the **10-year** profile? _____

Question 7

What is the **Water Surface Elevation** (m) at **R.K. 2.0** for each profile?

1-year: _____ 5-year: _____ 10-year: _____

Summary of Errors, Warnings, and Notes

The third type of output is the Summary of Errors, Warnings, and Notes text output. Click the icon for this feature and look at the messages for each profile. Note that there are many suggestions that more cross sections are needed.

Congratulations! You have completed the first HEC-RAS workshop.

Question 8

Optional Task: Close all output windows. From the main HEC-RAS menu, copy the project by selecting **Save Project As.** Give the new project the title: **"Test more cross sections"** and a new file name **"TESTINT.PRJ."** In the Geometric Data Editor, use the **XS Interpolation Tool** to see how close cross sections need to be to eliminate the warnings for more cross sections (Hint: try every 100 meters).

WEST CONSULTANTS, INC.

HEC-RAS WORKSHOP NO. 1

INTRO TO UNSTEADY FLOW MODELING

Objective: Develop a basic unsteady flow model in HEC-RAS.

Time to Complete: 60 minutes

Preface

Advanced HEC-RAS Workshops

Welcome to the first HEC-RAS workshop. We have tried to make these workshops as easy-to-follow and informative as possible. Each workshop is divided into a number of major steps, with detailed instructions provided for each step. In addition, the instructions are interspersed with Notes and Warnings, which provide additional guidance on completing the workshop or on using the HEC-RAS program in general. The symbols designating Notes and Warnings are shown below.

*The **Note** symbol represents useful additional information, such as an alternate way to perform a function in HEC-RAS.*

*The **Warning** symbol represents important or critical information.*

Geometric Data Editor

Also, note that when new concepts are introduced in the workshop, and the instructions say to click an icon on the toolbar, this icon is shown graphically in the left margin, like the example shown to the left.

Finally, questions are included in the workshops to test your understanding of HEC-RAS concepts. Questions are designated by the symbol shown below.

Question 1

*The **Question** symbol represents an opportunity to test your HEC-RAS knowledge.*

After each workshop, your instructor will go over the workshop solutions and answer any additional questions that you might have.

This workshop was originally developed through an extensive effort by the staff and engineers at the US Army Corps of Engineers Hydrologic Engineering Center (HEC), Davis, CA.

Intro

Introduction

In this workshop you will create a basic unsteady flow model in HEC-RAS.

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.

While these data are from an actual study, the model and results of this workshop do not represent current or future conditions of the river and may have been modified for demonstrative purposes.

Step 1

Open Existing RAS Project

Open the project "Unsteady_Intro.prj" with the title "Beaver Creek – Workshop 1" from the C:\workshops\Adv_RAS_Workshops\ directory or from the directory specified by the Instructor. A project with geometry has already been created for you. You will need to create the Unsteady Flow Data and an Unsteady Flow Plan in order to run the model.

Step 2

Create an Unsteady Flow File

A flow file must first be created for unsteady flow simulations, just like for RAS steady flow simulations. Open a new Unsteady Flow file by clicking **Edit**, and then **Unsteady Flow Data**. Save the Unsteady Flow file as **Unsteady Flow File for Beaver Creek**.

Step 3

Enter the Upstream Boundary Condition

We will specify a hydrograph at our upstream boundary and manually enter (as opposed to specifying a DSS file) these data.

(a) The upstream boundary is a flow hydrograph. Click in the first cell underneath the "Boundary Condition Type" column and in line with the "RS 5.99" row.

(b) Click the "Flow Hydrograph" button to open the Flow Hydrograph Editor.

(c) Select "Enter Table" (the circle may already be filled for you) and then "Fixed Start Time". Enter a date of "21May1994" and time of "2400". Set the "Data Time Value" to 1 hour.

(d) Enter the flow data listed in the table below or copy and paste the same data from the Excel file "Intro Boundary Condition Data.XLS" (located in the same directory as the RAS project).

Table 1. Beaver Creek Upstream Hydrograph

Date	Simulation Time (hours)	Flow (cfs)
21May1994 2400	00:00	500.00
22May1994 0100	01:00	730.00
22May1994 0200	02:00	1404.33
22May1994 0300	03:00	2477.03
22May1994 0400	04:00	3875.00
22May1994 0500	05:00	5502.97
22May1994 0600	06:00	7250.00
22May1994 0700	07:00	8997.03
22May1994 0800	08:00	10625.00
22May1994 0900	09:00	12022.97
22May1994 1000	10:00	13095.67
22May1994 1100	11:00	13770.00
22May1994 1200	12:00	14000.00
22May1994 1300	13:00	13770.00
22May1994 1400	14:00	13095.67
22May1994 1500	15:00	12022.97
22May1994 1600	16:00	10625.00
22May1994 1700	17:00	8997.03
22May1994 1800	18:00	7250.00
22May1994 1900	19:00	5502.97
22May1994 2000	20:00	3875.00
22May1994 2100	21:00	2477.03
22May1994 2200	22:00	1404.33
22May1994 2300	23:00	730.00
22May1994 2400	24:00	500.00

(e) Click the "Plot Data" button at the bottom of the Flow Hydrograph Editor to verify that your data look identical to Figure 1.

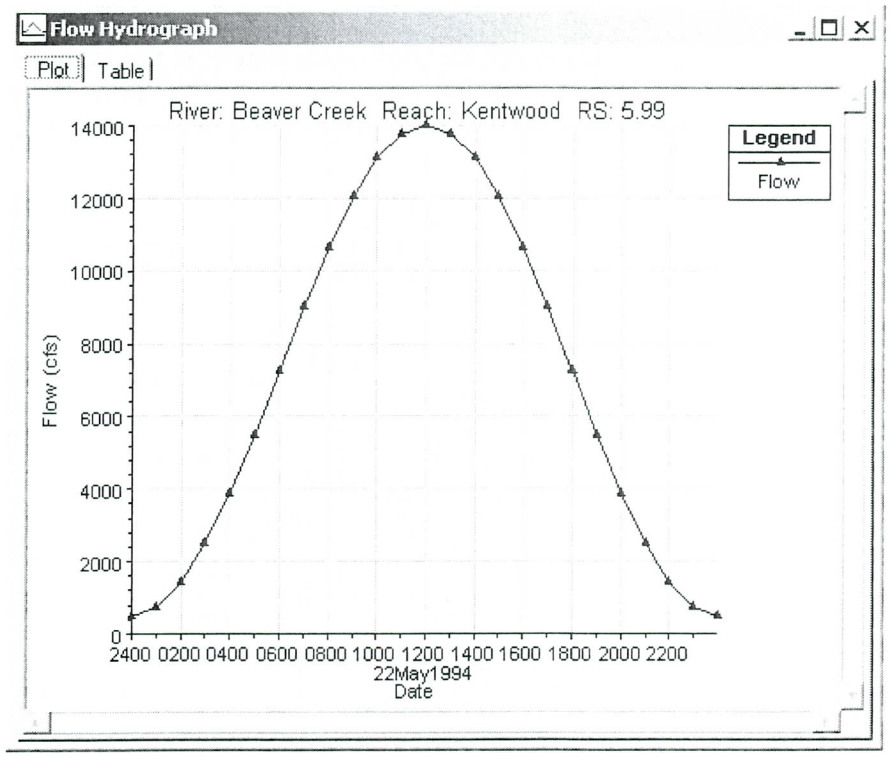


Figure 1. Upstream Boundary Condition Flow Hydrograph Plot

(f) Click the "OK" button when finished to close the Flow Hydrograph Editor. Keep the Unsteady Flow Data Editor open.

Step 4 Enter the Downstream Boundary Condition

You will now enter the downstream boundary, a rating curve, for the Beaver Creek project.

(a) Click in the second cell underneath the "Boundary Condition Type" column and in line with the "RS 5.00" row.

(b) Click the "Rating Curve" button to open the Rating Curve Editor.

(c) Enter the rating curve data in shown in the table below or copy and paste from the Excel file "Intro Boundary Condition Data.XLS" (located in the same directory as the RAS project).

Table 2. Beaver Creek Downstream Rating Curve.

Stage (ft)	Flow (cfs)
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

(d) Click the "Plot Data" button at the bottom of the Flow Hydrograph Editor to verify that your data look identical to Figure 2.

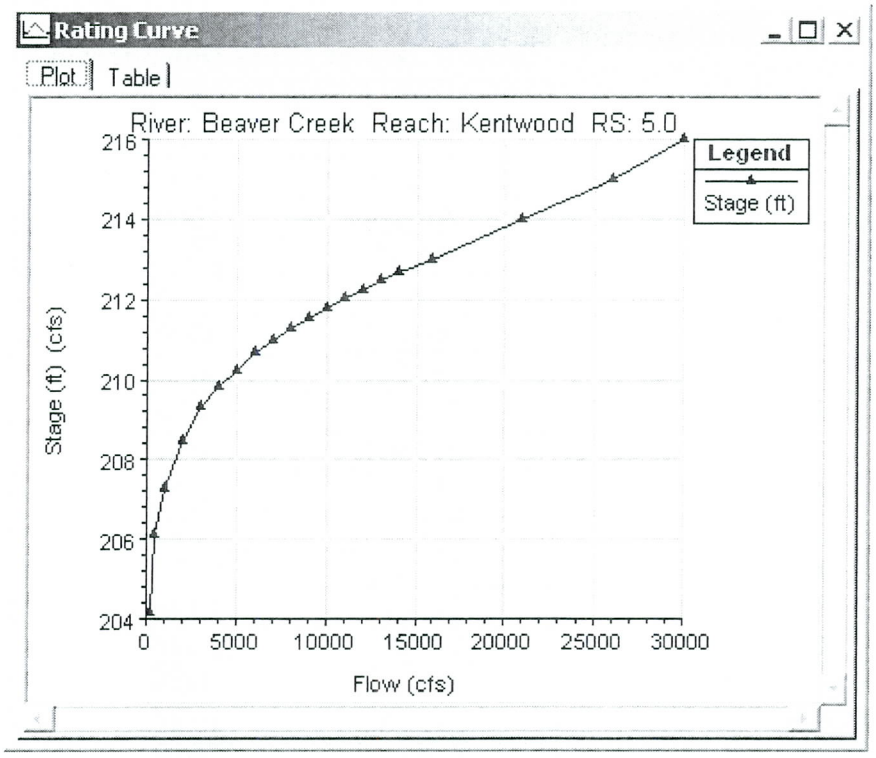


Figure 2. Downstream Boundary Condition Rating Curve Plot

(e) Click the "OK" button when finished to close the Rating Curve Editor. Keep the Unsteady Flow Data Editor open.

Step 5 Enter Initial Flow Data

RAS requires an initial flow at the upstream end of Beaver Creek.

(a) Select the "Initial Conditions" tab on the Unsteady Flow Data Editor.

(b) Click in the first cell underneath the "Initial Flow" column which is in line with the "RS 5.99" row. Enter a value of 500 cfs which is equal to the first discharge value in the upstream boundary condition flow hydrograph.

(c) Save the Unsteady Flow Data and close the Unsteady Flow Data Editor.

Step 6 Create an Unsteady Flow Plan

You now need to create an Unsteady Flow Plan in order to run the simulation.

(a) Open a new Unsteady Flow file by clicking **Run** and then **Unsteady Flow Analysis**.

(b) Select "Beaver Cr. Near Kentwood, Louisiana" as the Geometry File, and "Unsteady Flow File for Beaver Creek" as the Unsteady Flow File, if they are not already selected.

(c) Save the Unsteady Flow Data as "Beaver Creek Intro to Unsteady Flow" with a short plan identifying of "Intro".

(d) Select to run all three programs, "Geometry Preprocessor", "Unsteady Flow Simulation", and "Post Processor" by clicking in the box to the left of each label.

(e) Specify a start day of "21May1994" a start time of "2400", an end day of "22May1994", and an end time of "2400"

(f) Specify Computation Settings of 5 minutes for the "Computation Interval", and 1 hour for both the "Hydrograph Output Interval" the and "Detailed Output Interval". Save the Plan.

(g) Click the "Compute" button to run your simulation.

Step 7 Evaluate Results

Use the results from your unsteady flow run to answer the following questions:

Question 1 What is the peak discharge value at RS 5.0? What was the attenuation and lag in peak discharge between RS 5.99 and RS 5.0?

Question 2 Change the "Hydrograph Output Interval" to 5 minutes and view the results. Now change the "Hydrograph Output Interval" to 6 hours. Do these changes affect your answer to Question 1? Use the DSS feature (**View ➡ DSS Data** or the "DSS" button) to explain any differences.

Question 3

Use the various methods for viewing and animating RAS results (you may want to slow the animation speed down). How many different ways can you view animated results?

Question 4

What are the default hydrograph locations for this project? Add a hydrograph output location at RS 5.13, change the "Hydrograph Output Interval" back to 1 hour, and re-run your simulation.

Question 5

Change the "Computation Interval" to 2 hours and run the simulation. What occurs? Why?

Question 6

Change the "Computation Interval" to 1 hour in your new plan and then run the simulation. Don't be surprised if you get unrealistic results or the model does not converge on a solution. View the hydrograph at RS 5.13. Have the results changed? If so, how and why?

WEST CONSULTANTS, INC.

HEC-RAS WORKSHOP NO. 1

INTRO TO UNSTEADY FLOW MODELING

Question 1

What was the peak discharge value at RS 5.0? What was the attenuation and lag in peak discharge between RS 5.99 and RS 5.0?

This can be determined in numerous ways. The peak flow can be found by selecting **View** from the main RAS window, and then **Stage and Flow Hydrographs**. RS 5.0 can be selected from the drop down menu and the "Table" tab clicked to view results in tabular form (Figure 1). In this table the peak is **13890.59 cfs**, shown occuring on the 22nd of May at noon. Viewing the hydrograph plot also shows the same peak value (Figure 2).

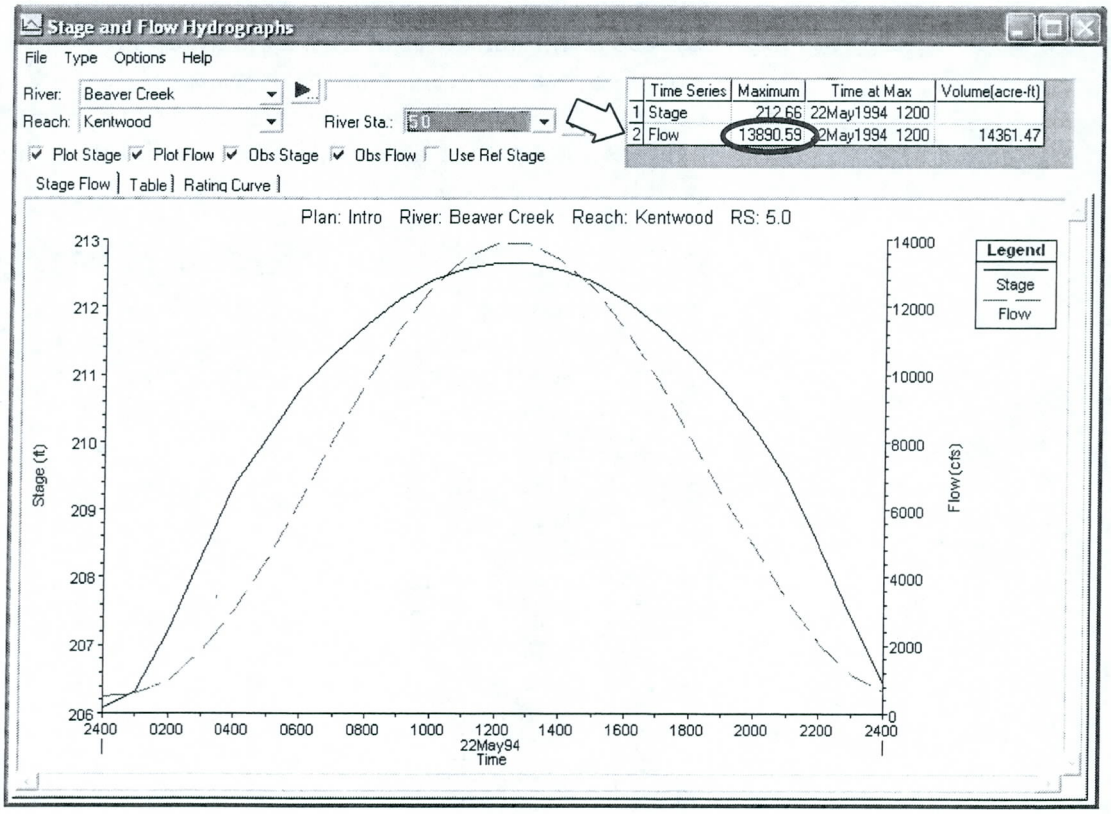


Figure 1

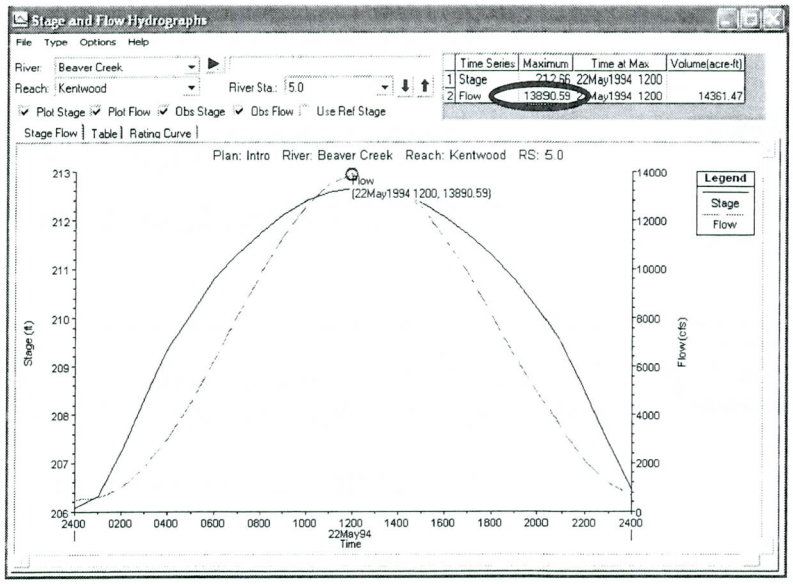


Figure 2

Another method is to go to the Detailed Output Table for RS 5.0 and select the "Max WS" profile. The flow associated with the "Max WS" is **13942.26 cfs** but it is not given a date or time (note that the maximum water surface elevaton may not always correspond with the maximum flow). The Summay Profile Tables show the "Max WS" value but this value is higher than any value associated with a time step in the table (Figure 3).

Profile Output Table - Standard Table 1

File Options Std. Tables User Tables Locations Help

HEC-RAS Plan: Intro River: Beaver Creek Reach: Kentwood (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	5.13	22MAY1994 2200	1947.21	199.90	209.23		209.40	0.001109	3.90	1182.09	575.52	0.28
Kentwood	5.13	22MAY1994 2300	1089.24	199.90	208.12		208.22	0.000801	2.91	708.23	315.10	0.23
Kentwood	5.1	22MAY1994 2400	636.42	199.90	207.14		207.21	0.000635	2.36	441.32	230.30	0.20
Kentwood	5.0	Max WS	13942.26	[illegible]	212.67	210.09	212.81	0.001203	4.88	8228.31	1973.39	0.31
Kentwood	5.0	21MAY1994 2400	500.50	199.90	206.09	202.94	206.16	0.001204	2.23	295.67	191.08	0.26
Kentwood	5.0	22MAY1994 0100	583.91	199.90	206.30	203.20	206.38	0.001268	2.37	341.07	235.58	0.26
Kentwood	5.0	22MAY1994 0200	945.92	199.90	207.21	204.33	207.29	0.001131	2.52	662.66	496.05	0.26
Kentwood	5.0	22MAY1994 0300	1842.23	199.90	208.33	205.82	208.41	0.001184	2.92	1315.21	648.99	0.27
Kentwood	5.0	22MAY1994 0400	2992.91	199.90	209.36	207.01	209.46	0.001119	3.21	2258.44	1287.92	0.27
Kentwood	5.0	22MAY1994 0500	4497.50	199.90	210.09	207.90	210.19	0.001154	3.62	3355.66	1729.39	0.29
Kentwood	5.0	22MAY1994 0600	6239.36	199.90	210.80	208.31	210.91	0.001153	3.96	4639.62	1851.07	0.29
Kentwood	5.0	22MAY1994 0700	8127.01	199.90	211.32	208.82	211.44	0.001178	4.24	5622.98	1890.02	0.30
Kentwood	5.0	22MAY1994 0800	9776.57	199.90	211.74	208.92	211.87	0.001189	4.45	6419.10	1920.97	0.30
Kentwood	5.0	22MAY1994 0900	11314.71	199.90	212.11	209.59	212.24	0.001193	4.62	7127.13	1948.11	0.31
Kentwood	5.0	22MAY1994 1000	12592.51	199.90	212.39	209.84	212.53	0.001195	4.75	7682.40	1962.66	0.31
Kentwood	5.0	22MAY1994 1100	13456.60	199.90	212.57	210.01	212.70	0.001203	4.84	8027.25	1969.44	0.31
Kentwood	5.0	22MAY1994 1200	13890.59	199.90	212.66	210.05	212.80	0.001203	4.88	8206.81	1972.97	0.31
Kentwood	5.0	22MAY1994 1300	13861.43	199.90	212.65	210.04	212.79	0.001203	4.88	8194.77	1972.73	0.31
Kentwood	5.0	22MAY1994 1400	13388.13	199.90	212.55	209.97	212.69	0.001203	4.83	7998.88	1968.89	0.31
Kentwood	5.0	22MAY1994 1500	12509.54	199.90	212.35	209.83	212.49	0.001213	4.77	7608.04	1961.19	0.31
Kentwood	5.0	22MAY1994 1600	11278.74	199.90	212.07	209.59	212.20	0.001218	4.65	7058.00	1945.46	0.31

Figure 3

Comparing hydrographs at RS 5.99 and RS 5.0 and determining the attenuation and lag in the peak discharge, can be determined by using the DSS feature (**View** then **DSS Data** or the "DSS" button). The results indicate that there is an attenuation of approximately 109.4 cfs and no lag (when the output interval is set to 1 hour). Time series values can be revealed by clicking on points on the graph or by switching to the "Table" tab (Figure 4 and Figure 5). Time series values can also seen by selecting all "profiles" in the Summary Tables.

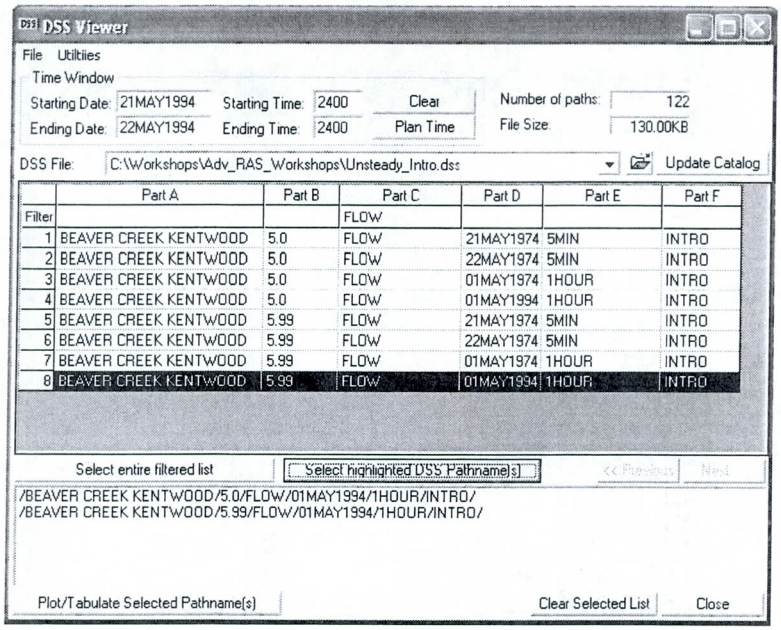


Figure 4

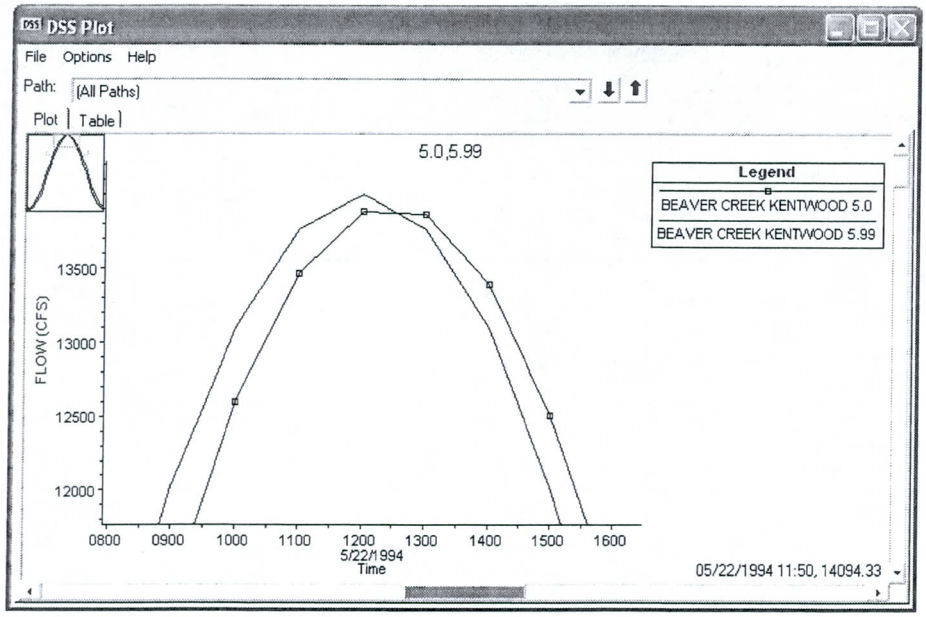


Figure 5

Question 2

Change the "Hydrograph Output Interval" to 5 minutes and view the results. Do these changes affect your answer to Question 1? Consider using the DSS feature to explain any differences.

By changing the output interval to 5 minutes the hydrograph is defined by more points. This affects the reported peak values. The attenuation is approximately 57.74 cfs and lag approximately 20 minutes <u>for results generated with the Hydrograph Output Interval set to 5 minutes</u> (Results from DSS shown in Figure 6). Figure 7 shows the difference in DSS between setting the output interval at 5 minutes and 1 hour.

DSS Plot

File Options Help

Path: (All Paths)

Plot Table

		BEAVER CREEK KENTWOOD 5.99	BEAVER CREEK KENTWOOD 5.0
	Date	INST-VAL	INST-VAL
		CFS	CFS
140	22May1994 11:35	13904.17	13767.19
141	22May1994 11:40	13923.33	13795.25
142	22May1994 11:45	13942.50	13821.17
143	22May1994 11:50	13961.67	13845.46
144	22May1994 11:55	13980.83	13868.50
145	22May1994 12:00	14000.00	13890.59
146	22May1994 12:05	13980.83	13910.69
147	22May1994 12:10	13961.67	13926.72
148	22May1994 12:15	13942.50	13937.30
149	22May1994 12:20	13923.33	13942.26
150	22May1994 12:25	13904.17	13942.16
151	22May1994 12:30	13885.00	13937.81
152	22May1994 12:35	13865.83	13930.05
153	22May1994 12:40	13846.67	13919.60
154	22May1994 12:45	13827.50	13907.11
155	22May1994 12:50	13808.33	13893.01
156	22May1994 12:55	13789.17	13877.70
157	22May1994 13:00	13770.00	13861.43
158	22May1994 13:05	13713.81	13843.23
159	22May1994 13:10	13657.61	13821.14
160	22May1994 13:15	13601.42	13793.80

Figure 6

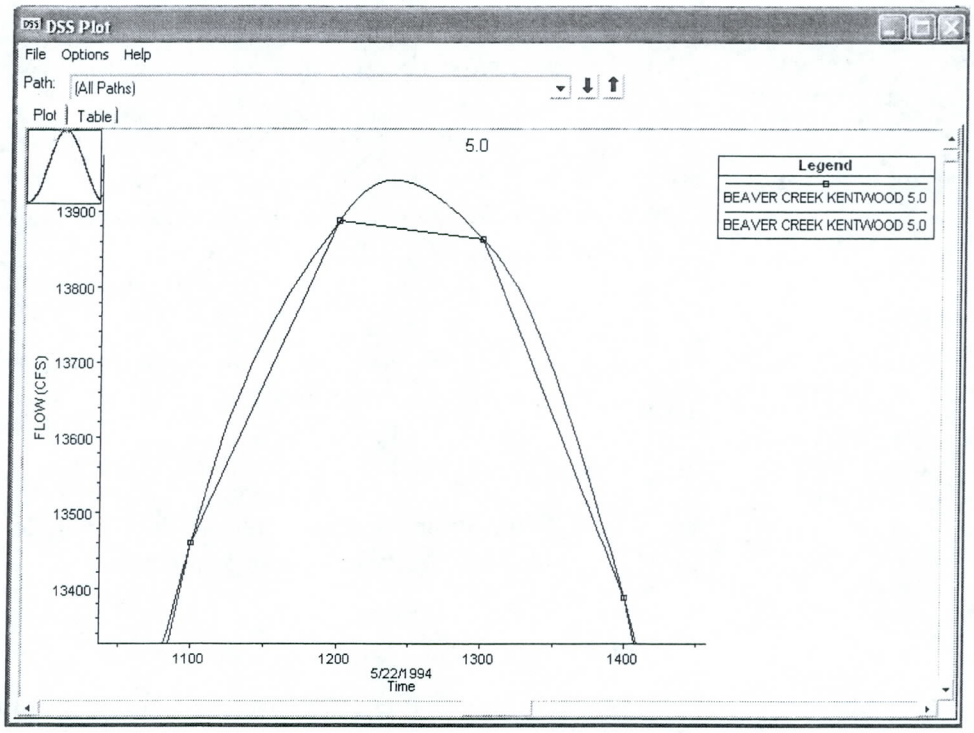


Figure 7

Notes on selecting computation and output intervals: When the user changed the output intervals, only the time step at which results can be view is affected. Computed results are independent of the output interval selected by the user.

Increasing the output interval decreases the computation time (and reduces the size of the DSS output and animation files); however, we've seen that this reduces the resolution of the results. For this single reach project the difference in computation time is probably less than one second (depending on your computer processing speed) when the output interval varies between 5 minutes and 1 hour, but this can have a larger effect on complex projects. Decreasing the "Detailed Output Interval" vs. "Detailed Hydrograph Output" typically has a greater impact on the computation time. The output intervals can be chosen to minimize computation time while still maintaining adequate resolution in the generated results.

Question 3

Get familiar with the various methods for viewing RAS results and animating results (you may want to slow the animation speed down). How many different ways can you view animated results?

Animation can be viewed on cross section plots, the profile plots, the X-Y-Z perspective plot, and the general profile plot (e.g., velocity vs. distance).

Question 4 *What are the default hydrograph locations for this project?*

Hydrographs are generated at the upstream and downstream boundaries, RS 5.99 and 5.0, by default. Additional hydrograph locations can be specified from **Stage and Flow Output Locations** under **Options** on the Unsteady Flow Analysis Editor.

Question 5 *Create a new plan. Change the "Computation Interval" to 2 hours and run the simulation. What occurs? Why?*

Multiple errors occur:

1) the "Hydrograph Output Interval" (= 1 hr) is less than the "Computation Interval" (= 2 hrs), and

2) the "Detailed Output Interval" (=1 hr) is less than the "Computation Interval" (=2 hrs).

This is one example of checks RAS makes of the input data. The reason these errors occurred are that the output intervals are not allowed to be less than the computation interval (i.e., results will not be interpolated between computed points in time).

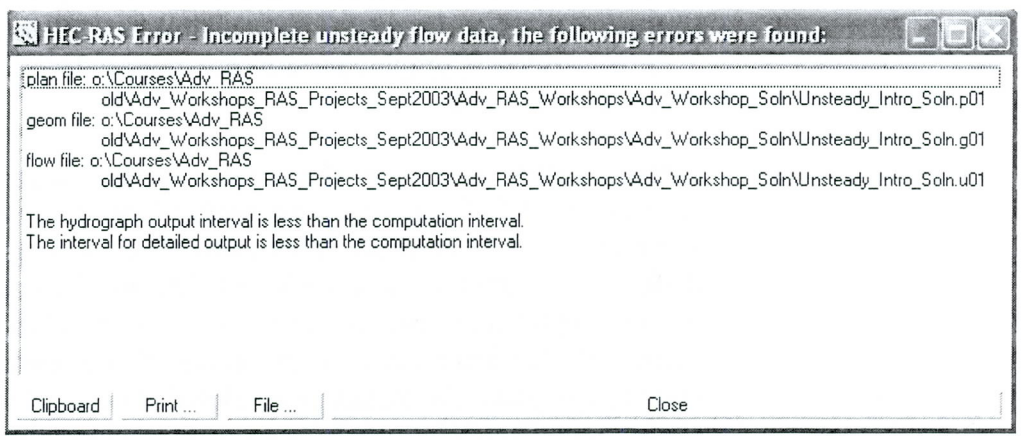


Figure 8

Question 6 *Change the "Computation Interval" to 1 hour in your new plan, and run the simulation. Have the results changed? If so, how and why?*

The model is unstable when set to compute at 1 hour time steps (tips on creating a stable model and selecting appropriate time steps are discussed in later lectures). The first indication is the "Maximum iterations" computation messages in the Hydraulic Computation window. Evidence of the instability can be seen in the fluctuating stage and flow (Figure 9 and Figure 10).

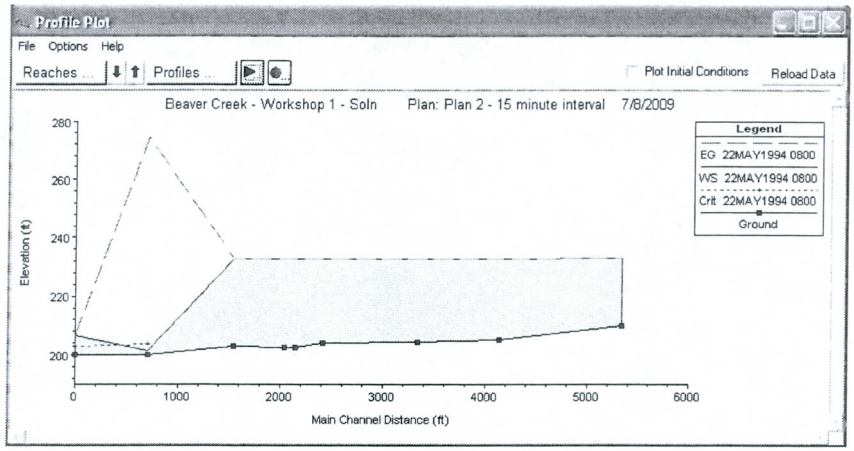


Figure 9

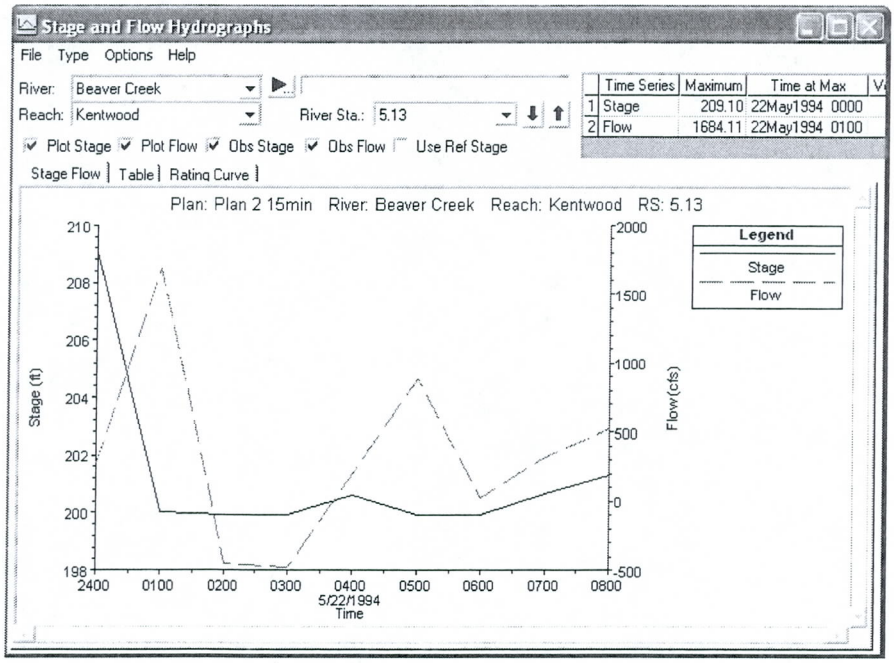


Figure 10

WEST CONSULTANTS, INC.

HEC-RAS Workshop No. 2

Adding Tributaries and Junctions

Objective: Learn how to create a river network by adding tributaries and junctions.

Time to Complete: 45 minutes

Intro

Steps for Workshop No. 2

In the second workshop, we are going to add Cyrus Wash, a Kern River tributary, expand the Kern River System by adding the Upper Kern River, and add a junction where Cyrus Wash and the Upper Kern River join the Lower Kern River.

This workshop is divided into seven steps:

Step 1. Copy and rename a previous project into a different directory

Step 2. Create a river network

Step 3. Enter junction data

Step 4. Create cross sections on Upper Kern River

Step 5. Create cross sections on Cyrus Wash

Step 6. Enter steady flow data

Step 7. Run steady flow analysis, view results, and answer questions

Step 1

Copy and Rename a Previous Project into a Different Directory

For this workshop, we will start with the HEC-RAS project developed in Workshop No. 1. To ensure that everyone is starting with the same project, you will be starting with a copy of the Workshop No. 1 solution file.

Open Project

(a) From the File menu, select **Open Project**.

(b) Go to the **C:\Workshops\Basic_RAS_Workshops\Basic_Workshop_Soln** directory by double-clicking on the appropriate folders on the right side of the window.

(c) Select the project titled **Developing a Hydraulic Model (Solution)**, with the file name **Wk1_Develop_Soln.prj**. Click **OK**.

Now, we want to make a copy of this project to start Workshop No. 2.

(d) From the File menu, select **Save Project As**.

(e) Double-click on **C:\Workshops\Basic_RAS_Workshops** in the directories window. Click the **Create Folder** button. Type **Workshop_2** for the name of the project directory.

(f) Type **Adding Tributaries and Junctions** as the title.

(g) Enter **Wk2_AddTribs.prj** as the project file name, then click **OK**.

*When you tab over to the File Name entry box, the default file name extension, .prj, is already present. Don't forget to delete the * after typing the file name.*

You will be returned to the HEC-RAS main menu screen.

(h) Update the description in the Project Description window (e.g., **Workshop No. 2 – Adding Tributaries and Junctions**).

Save Project

(i) Click the **Save Project** icon on the toolbar.

Step 2 Create a River Dendritic System

We want to add a new reach – the Upper Kern River – on the Kern River system, and a new river (Cyrus Wash) with one reach (Lower Cyrus) to create the river network. The network will also included one junction.

Geometric Data Editor

(a) Click the **Geometric Data** editor icon on the toolbar.

*If the entire reach is not displayed on the schematic, select **Set Schematic Plot Extents** from the View menu. Click the **Set to Computed Extents** button, and then click **OK** to update the schematic.*

Before we draw the new reaches, we need to make room for them on the schematic by moving the Lower Kern River to the lower right section of the window.

(b) From the Edit menu of the Geometric Data editor, select **Move Object**.

This will add crosshairs to your cursor, which allows you to move points on the reach, as well as the reach names.

The HEC-RAS program doesn't allow you to select and move an entire reach at once. You have to move the reach by moving its points individually.

(c) Move the river reach to the lower right section of the window by clicking and dragging the points next to R.K. 2.0 and R.K. 1.0, if necessary. Drag and drop the reach label (Lower Kern River) next to the new reach location.

While the reach label ("Lower Kern River") can be moved, the river label ("Kern R. System") and river direction arrow can't be moved manually; they move automatically with the river schematic.

(d) From the Edit menu, select **Move Object** a second time, which will turn off this option.

Now we're ready to add the Upper Kern River.

(e) Click the **River Reach** icon. Move the cursor – which will now look like a pencil – to the top left corner of the window.

(f) Click the left mouse button once, move the pencil to cross section 2.0 on the Lower Kern River, and then double-click the left mouse button.

This will bring up a window asking for the name of the river and reach (with default names shown).

(g) For the River name, use the default **Kern R. System**. For the Reach, enter **Upper Kern River**. Click **OK**.

You will then be prompted to name the junction connecting the Upper Kern River to the Lower Kern River.

(h) Enter **Bear Junct.** for the junction name.

Next, we will add Cyrus Wash, a Kern River tributary that enters at Bear Junction.

(i) Click the **River Reach** icon, and then move the cursor to the right of top center on the schematic drawing of the Kern River System.

(j) Click the left mouse button once, move the pencil to cross section 2.0 on the Lower Kern River, and then double-click the left mouse button.

(k) For the river name, enter **Cyrus Wash**. For the reach, enter **Lower Cyrus**. Click **OK**.

(l) From the File menu, select **Save Geometry Data**.

Step 3 Enter Junction Data

Now we need to enter additional data for the junction.

(a) Click the **Junction** button on the left side of the window, which opens the Junction Data editor.

(b) For the description, enter **Confluence of Upper Kern River and Cyrus Wash with the Lower Kern River**.

The program allows junction computations to use the energy or momentum method. Because Cyrus Wash comes into the Kern River system at a significant angle, we want to use the momentum method.

(c) Select **Momentum** for the Computation Mode. Leave the default **Add Friction** selected.

(d) For Length (m) across the junction, enter **30** from the Lower Kern River to each reach. For Tributary Angle (Deg), enter **0** for the angle from Lower Kern River **to Upper Kern River**. Enter **45** for the angle from Lower Kern River to **Cyrus Wash - Lower Cyrus**. The completed table is shown below.

Length across Junction		Tributary Angle (Deg)
From: Kern R. System - Lower Kern River	Length (m)	
To: Kern R. System - Upper Kern River	30	0
To: Cyrus Wash - Lower Cyrus	30	45

(e) Click **Apply Data**, and then click **OK**.

(f) From the File menu, select **Save Geometry Data**.

Step 4 Create Cross Sections on Upper Kern River

We are going to create two cross sections on the Upper Kern River. First, we will copy R.K. 2.0 on Lower Kern River to create R.K. 2.03 on the Upper Kern River.

(a) Open the **Cross Section Data** editor

(b) Select **Kern R. System** for the River, **Lower Kern River** for the Reach, and **2.0** for the River Station.

(c) From the Options menu, select **Copy Current Cross Section**.

You will prompted for the River and Reach to which the cross section will be copied.

(d) Leave **Kern R. System** as the River. Select **Upper Kern River** for the Reach.

*Make sure that you select **Upper Kern River** for the Reach. If **Lower Kern River** is mistakenly selected, the new cross section will be added to the wrong reach.*

(e) Enter **2.03** for the new river station, then click **OK**.

You will be returned to the Cross Section Data editor with the Upper Kern River and R.K. 2.03 showing.

(f) Change the description to **River Kilometer 2.03 on Upper Kern River**.

(g) Enter the following **Downstream Reach Lengths (m)**:

LOB	Channel	ROB
0	0	0

The downstream reach lengths for the first cross section in each reach above the junction are set to zero because reach lengths across the junction are input in the Junction Data Editor.

(h) From the Options menu, select **Adjust Stations** and **Multiply by a Factor**.

(i) Enter **0.8** (20% decrease) for the LOB , **0.8** for the Channel, and **0.8** for the ROB. When finished, click **OK** or press enter.

(j) On the Cross Section Data editor, click **Apply Data**.

(k) From the File menu, select **Save Geometry Data**.

Next, we want to add a cross section at River Kilometer 2.5, which will be a modified copy of R.K. 2.03.

(l) Return to the **Cross Section Data** editor and confirm that River Sta. **2.03** on the **Upper Kern River** is shown.

(m) From the Options menu, select **Copy Current Cross Section**.

(n) Confirm that **Kern R. System** is selected as the River, and **Upper Kern River** as the Reach.

When copying a cross section to the same reach, the correct River and Reach will already be selected.

(o) Enter **2.5** for the new river station, then click **OK**.

(p) Change the description to **River Kilometer 2.5 on Upper Kern River**.

(q) From the Options menu, select **Adjust Elevations**. Enter **1.5**, and then click **OK**.

(r) Enter the following **Downstream Reach Lengths (m)**:

LOB	Channel	ROB
470	470	470

(s) Click **Apply Data**.

(t) From the Plot menu, select **Plot Cross Section** and check the new cross sections. Also from the Plot menu, select **Plot Profile** to check that the profile looks correct. When finished, close the plots.

(u) From the File menu, select **Save Geometry Data**.

Step 5 Create Cross Sections on Cyrus Wash

We are also going to create two cross sections on Cyrus Wash, a Kern River tributary. Both of these cross sections will be on the Lower Cyrus reach.

(a) Open the **Cross Section Data** editor.

(b) Change the River to **Cyrus Wash**, and the Reach to **Lower Cyrus**.

(c) From the Options menu, select **Add a New Cross Section**.

(d) Enter **0.03** for the new river station, then click **OK**.

(e) In the Description box, enter **River Kilometer 0.03 on Lower Cyrus**.

(f) Enter the following **Cross-Section X-Y Coordinates** (Station-Elevation data in meters):

	Station	Elevation
1	3.0	5.2
2	5.5	4.3
3	7.3	2.3
4	8.5	2.4
5	9.8	4.6
6	15.2	5.8

(g) Enter the following **Downstream Reach Lengths (m)**:

LOB	Channel	ROB
0	0	0

(h) Enter the following **Manning's n Values**:

LOB	Channel	ROB
0.05	0.03	0.06

From field inspection and other evaluations, we have determined the location of the channel bank stations.

(i) Enter the following **Main Channel Bank Stations (m)**.

Left Bank	Right Bank
5.5	9.8

(j) Click **Apply Data**.

The cross section at R.K. 0.8 will be based on a copy of the R.K. 0.03 cross section.

(k) From the Options menu, select **Copy Current Cross Section**.

(l) Confirm that **Cyrus Wash** is selected as the River, and **Lower Cyrus** as the Reach.

(m) Enter **0.8** for the new river station, then click **OK**.

(n) Change the description to **River Kilometer 0.8 on Lower Cyrus**.

(o) From the Options menu, select **Adjust Elevations**. Enter **0.5**, then click **OK**.

(p) Enter the following **Downstream Reach Lengths (m)**:

LOB	Channel	ROB
770	770	770

(q) Click **Apply Data**.

(r) From the Plot menu, select **Plot Cross Section** and then **Plot Profile** to check that the cross sections and profile look correct. When finished, close the plots and Cross Section Data editor.

(s) From the File menu, select **Save Geometry Data**.

(t) Close the Geometric Data editor, leaving the main HEC-RAS window.

Save Project

(u) Save the project (click the **Save Project** icon on the toolbar).

Step 6 Enter Steady Flow Data

As in Workshop No. 1, we want to run three subcritical flow profiles (1-year, 5-year, and 10-year discharges).

Steady Flow Data Editor

(a) Click the **Steady Flow Data** editor icon on the toolbar.

(b) Confirm that **3** is shown in the **Enter/Edit Number of Profiles (25,000 max)** box.

Discharge values for the Lower Kern River were entered during Workshop No. 1, and will be left unchanged. However, discharges are required at the upstream end of the Upper Kern River and Lower Cyrus for each profile.

(c) Enter the following flow rates (m^3/s):

River	Reach	RS	1-year	5-year	10-year
Kern R. System	Lower Kern River	2.0	3	15	30
Kern R. System	Upper Kern River	2.5	2	11	24
Cyrus Wash	Lower Cyrus	0.8	1	4	6

(d) From the File menu, select **Save Flow Data**.

Because we are running subcritical flow, we only need to specify reach boundary conditions at the most downstream cross section of the river system, namely R.K. 1.0 on the Lower Kern River. Boundary conditions at this cross section were previously specified for each of the three profiles, and will be left unchanged.

(e) Click the **Reach Boundary Conditions** button and examine the boundary conditions table.

Note that internal boundary conditions are automatically created for the junction. These boundary conditions cannot be edited by the user.

Question 1 How does HEC-RAS compute the starting water surface elevations at R.K. 0.03 on Lower Cyrus, and R.K. 2.03 on the Upper Kern River?

_____

(f) Close the Steady Flow Data window and return to the main HEC-RAS window. Save the project.

Step 7 Run Steady Flow Analysis, View Results, and Answer Questions

Steady Flow Analysis

(a) Return to the Steady Flow Analysis window, and click the large **Compute** button at the bottom.

(b) When computations are finished, click the **Close** button on the progress window.

(c) Close the Steady Flow Analysis window to return to the main HEC-RAS window.

Question 2 Using the profile and/or cross section output tables, fill out the following table for water surface elevations and top widths (hint: include profile name in the profile output table):

Reach	River Sta	Profile	W.S. Elev (m)	Top Width (m)
Upper Kern River	2.5	5-year		
Lower Kern River	2.0	1-year		
Lower Cyrus	0.8	5-year		

Question 3 At what cross sections, reaches, and profiles does the flow go critical?

_____

Next, we want to look at a profile plot showing both the Kern River System and Cyrus Wash.

W.S. Profile Plot

(d) Click the **Water Surface Profiles** icon on the toolbar to open the profile plot window.

(e) Click the **Reaches...** button below the menu bar. Select only the **Lower Kern River** and **Upper Kern River** reaches.

(f) From the Options menu, select **Land Marks** and then **River Stationing**.

(g) Look at invert and slope of each reach.

(h) Repeat tasks (e) through (g) for the **Lower Kern River** and **Lower Cyrus**.

Question 4 Based on the profile plots, would critical depth results be reasonable if found on Cyrus Wash? Why or why not? _____

_____

Question 5 Based on the profile plots, would critical depth results be reasonable if found on Upper Kern River? Why or why not?_____

_____

Question 6 Examine the cross sections of Cyrus Wash compared to the computed water surfaces. What could you do to improve the model?_____

_____

*If the water surface elevations are difficult to see on the plot, you may have too many variables showing. From the Options menu, select **Variables**. As necessary, turn off **Critical Depth**, **Energy Grade**, or other variables.*

Congratulations! You have completed the second HEC-RAS workshop.

Question 7

Optional Task: What is the warning for the 1-year profile at the first cross section upstream of the junction for each reach? What could you do to fix this? _____

_____

_____

WORKSHOP SOLUTIONS

WEST CONSULTANTS, INC.

HEC-RAS WORKSHOP NO. 2

ADDING TRIBUTARIES AND JUNCTIONS

Question 1

How does HEC-RAS compute the starting water surface elevations at R.K. 0.03 on Lower Cyrus, and R.K. 2.03 on the Upper Kern River?

For subcritical flow – flow combining, backwater calculations are performed separately across the junction for each of the upstream reaches. The backwater calculations are applied separately from R.K. 2.0 of the Lower Kern River to R.K. 0.03 on Lower Cyrus and to R.K. 2.03 on the Upper Kern River. The water surface elevations in this case are determined with the momentum equation [Hydraulic Reference Manual, pages 4-9 and 4-15].

Question 2

Using the profile and/or cross section output tables, fill out the following table for water surface elevations and top widths.

Reach	River Sta	Profile	W.S. Elev (m)	Top Width (m)
Upper Kern River	2.5	5-year	5.52	22.96
Lower Kern River	2.0	1-year	3.96	28.24
Lower Cyrus	0.8	5-year	5.07	5.04

Question 3

At what cross sections, reaches, and profiles does the flow go critical?

R.K. 2.5 *on the* ***Upper Kern River*** *during the* ***5-year*** *discharge [Cross Section or Profile Output Table]*

You can also find where critical depth is the solution by opening the ***Summary of Errors, Warnings, and Notes****, and selecting* ***All Rivers*** *and* ***All Profiles****.*

Although critical depth values may be displayed in an output table, critical depth is not the solution unless it is equal to the computed water surface elevation. Also note that the program may default to critical depth because it can't reach a solution due to an insufficient number of cross

This document is property of WEST Consultants, Inc., and is not to be duplicated without prior consent.

sections, errors caused by bad input data, wrong flow regime selected, and inability to balance the water surface elevation near the top of levees or ineffective flow areas [User's Manual page 11-8, Hydraulic Reference Manual page 2-13].

Question 4

Based on the profile plots, would critical depth results be reasonable if found on Cyrus Wash? Why or why not?

Since Lower Kern River has subcritical flow and Cyrus Wash has about the same slope with a lower discharge and corresponding cross section size, we would be concerned if Cyrus Wash went critical [Profile Plot].

Question 5

Based on the profile plots, would critical depth results be reasonable if found on Upper Kern River? Why or why not?

Since the Upper Kern River has a significantly steeper slope, it could go critical if it had very coarse bed material or bedrock [Profile Plot].

Question 6

Examine the cross sections of Cyrus Wash compared to the computed water surfaces. What could you do to improve the model?

The water surface elevation for the 10-year profile was higher than the left end point of the cross section for R.S. 0.03. This end should be extended using available topographic information so that the cross section totally "contains" the water surface of the highest discharge. Note that a warning stating that the cross section end points were extended is given in the Cross Section Output table for R.S. 0.8 [Cross Section Plots, Cross Section Output Table, and Summary of Errors, Warnings, and Notes].

Question 7

Optional Task: What is the warning for the 1-year profile at the first cross section upstream of the junction for each reach? What could you do to fix this?

The warning states that the momentum calculations at the junction produced a lower upstream energy than downstream. The upstream energy was set to the downstream energy.

To fix this, one could create a new geometry file for the 1-year profile and run the junction calculations with the energy equation in a separate plan.

This document is property of WEST Consultants, Inc., and is not to be duplicated without prior consent.

WEST CONSULTANTS, INC.

HEC-RAS Workshop No. 3

Creating Plans

Objective: Learn how to create and compare unique Plans with different geometric and hydrologic conditions within the same Project.

Time to Complete: 50 to 60 minutes

Preface

General Information

The project conditions include a proposed levee on the left side of the Lower Kern River from R.K. 1.0 to R.K. 2.0 and a proposed detention basin upstream of the Upper Kern River. The levees are intended to contain the river flow from the left overbank for each profile and the detention basin will reduce the peak discharges in the river. Since either one or both of these projects may be required, we are asked to determine the hydraulic effects for various cases.

The required analyses are:

Plan 1 – Existing (base) conditions, comprised of existing geometry and existing conditions steady flow data

Plan 2 – Levee geometry with existing conditions steady flow data

Plan 3 – Existing conditions geometry with detention basin steady flow data

Plan 4 – Levee geometry with detention basin steady flow data

Intro

Steps for Workshop No. 3

The third HEC-RAS workshop – Creating Plans – is divided into five steps:

Step 1. Copy and rename a previous project into a different directory

Step 2. Enter project conditions (levee) geometric data

Step 3. Enter project conditions (detention basin) steady flow data

Step 4. Combine data files into project plans and run steady flow analysis

Step 5. View results and answer questions

Step 1 Copy and Rename a Previous Project into a Different Directory

(a) From the File menu, select **Open Project**.

(b) Browse for and then double-click on **C:\Workshops\Basic_RAS_Workshops** in the directories window. Double click on the **Basic_Workshop_Soln** folder.

(c) Double click on **Wk2_AddTribs_Soln.prj** to open the file.

An alternate way to open a file is to check to see if it is listed at the bottom of the File menu. The last 8 projects that have been worked on in HEC-RAS are shown. You can click on any one of those files to open them.

(d) From the File menu, select **Save Project As**.

(e) Double-click on **C:** in the directories window. Click on the **Create Folder** button. Type **Workshop_3** for the name of the new project directory. Click the **OK** button.

(f) Delete the current title and type **Creating Plans** as the title. Tab over to File Name window and type **Wk3_CreatePlans** for the project file name and click **OK**.

*When you tab over to the File Name entry box, the default file name extension, .prj, is already present. Don't forget to delete the * after typing the file name.*

You will be returned to the HEC-RAS main menu screen.

(g) Delete the current project description and enter a suitable description (e.g., **Workshop No. 3 - Creating Plans**) in the Project Description window.

Save Project

(h) Click the **Save Project** icon on the toolbar.

Step 2 Enter Project Conditions (Levee) Geometric Data

Geometric Data Editor

(a) Open the Geometric Data editor by clicking the **Geometric Data** editor icon on the toolbar.

(b) From the File menu, select **Save Geometry Data As**. Delete the current title and type **Levee on Lower Kern River**. Click **OK**.

*Remember that it is only necessary to provide a title; the file name is created by the HEC-RAS program in sequential order (i.e. *.g02, etc.)*

(c) Click the **Cross Section** button on the left side of the window, which opens the Cross Section Data editor. Toggle, if necessary, to River: **Kern R. System**, Reach: **Lower Kern River**, River Sta.: **1.0**.

Alternately, using the river system schematic, position the cursor at R.K. 1.0 of the Lower Kern River and click the left mouse button (you may need to try this several times by moving the cursor). When you get a window titled "Kern R. System Lower Kern River 1.0", click on ***Edit Cross Section****.*

(d) From the Options menu, select **Levees**. Enter **30** for the station and **4.6** for the elevation under the **Left** window (these are understood by the program to be in meters) and then click **OK**.

The bottom of the Cross Section Data editor window should now say "Levee on Left Side".

(e) Click the **Apply Data** button to bring the new information into program memory.

(f) Plot the cross section to check if the levee was placed properly. When finished, close the plot to return to the Cross Section Data editor.

(g) Click the up arrow next to the River Station to bring up R.K. **1.5**. Place a left levee at station **30** m with elevation **4.6** m in the same manner.

(h) Continuing for R.K. **2.0**, place a left levee at station **36** m and elevation **5.6** m. Click **Apply Data**.

(i) Check all the input on the cross section plots.

(j) Close the Cross Section Data editor. Save the geometric data by selecting **Save Geometry Data** under the File menu on the Geometric Data editor.

(k) Close the Geometric Data editor.

Step 3 Enter Project Conditions (Detention Basin) Steady Flow Data

We need to enter project conditions steady flow data for the three subcritical flow profiles (1-year, 5-year, and 10-year discharges). The 1-year profile has a known water surface elevation for the downstream boundary condition. The 5-year and 10-year profiles use normal depth for the downstream boundary condition.

Steady Flow Data Editor

(a) Click the **Steady Flow Data** editor icon on the toolbar.

(b) From the File menu, select **Save Flow Data As**. Delete the current title and type: **1, 5, and 10-year Qs with Detention Basin**. Click **OK**.

(c) Enter the following flow rates (m^3/s) for the three profiles:

River	Reach	RS	1-year	5-year	10-year
Cyrus Wash	Lower Cyrus	0.8	1	4	6
Kern R. System	Upper Kern River	2.5	0.3	7	14
Kern R. System	Lower Kern River	2.0	1.3	11	20

(d) Click the **Reach Boundary Conditions** button.

(e) Select **Set boundary for one profile at a time at** the top of the window.

This will add rows for all three profiles. Because we are modeling a subcritical profile, we need to add boundary condition data in the Downstream column only, and only for the Lower Kern River.

(f) Click on the cell for the **Lower Kern River**, **1-year** profile, below the **Downstream** column heading (the cell should be highlighted in blue).

(g) Click the **Known W.S.** button, and then enter **2.4** for the downstream known water surface elevation. Click **OK**.

(h) Click on the cell for the **Lower Kern River**, **5-year** profile, below the **Downstream** column heading.

(i) Click the **Normal Depth** button, and then enter **.001** for the slope. Click **OK**.

(j) Click on the cell for the **Lower Kern River**, **10-year** profile, below the **Downstream** column heading.

(k) Click the **Normal Depth** button, and then enter **.001** for the slope. Click **OK**. Click **OK** again to close the Steady Flow Boundary Conditions window.

(l) From the File menu, select **Save Flow Data**. Close the Steady Flow Data window and return to the main HEC-RAS window.

Step 4 Combine Data Files into Project Plans and Run Steady Flow Analysis

From the previous workshop, you already have a Plan with the short ID **Existing**. We need to create plans for the other conditions that are to be analyzed by combining the appropriate geometric and flow data files.

Steady Flow Analysis

(a) Open the Steady Flow Analysis window by clicking the **Steady Flow Analysis** icon on the toolbar.

The last saved geometry and steady flow data files will be listed.

(b) From the File menu, select **New Plan**.

If you get the message "Do you want to save existing plan data?", click **No**.

(c) Enter **Plan 2, Levee with Existing Qs**, for the title, and then click **OK**.

(d) Enter **Levee_exQ** for the short plan identifier (short ID) and click **OK**.

(e) Toggle the Geometry Files to select **Levee on Lower Kern River**

(f) Toggle the Steady Flow Files to select **1, 5, 10 yr Q's, Existing Conditions.**

(g) For the flow regime, click **Subcritical** (it may already be selected).

(h) From the Options menu, select **Critical Depth Output Option**, then select **Critical Always Calculated**.

(i) From the File menu, select **Save Plan**.

(j) Hit the large **Compute** button at the bottom of the Steady Flow Analysis window to run the plan.

(k) When computations are finished, click the **Close** button on the computation progress window.

(l) In a similar manner, create, save, and run **Plan 3, Existing Geom. with Detention Qs** (short ID: **ExGeom_dbQ**), and then **Plan 4, Levee with Detention Qs** (short ID: **Levee_dbQ**).

(m) Close the Steady Flow Analysis window, return to the main HEC-RAS window and select **Save Project** under the File menu.

Step 5 View Results and Answer Questions

Now we're ready to look at output from the runs.

Cross Section

(a) Click the **Cross Section** icon on the toolbar to open the cross section graphical output window.

(b) From the Options menu, select **Profiles**. Click the **Select All** button to show all profiles in the plot, and then click **OK** to confirm.

(c) Also from the **Options** menu, select **Plans** to make your selection of Plans to look at.

Note that when you plot the cross sections, it will show the cross section geometry of the last Plan that you opened in the Steady Flow Analysis window. You can change the geometry shown by going to the Geometry Data editor window that shows the dendritic schematic of the system and opening the geometry file you want to have displayed.

The plots of the water surface elevations are at the correct elevation for the last Plan opened, but the lateral limits in the figure may not be accurate, depending on the geometry file that is shown.

Profile Table

(d) From the HEC-RAS main screen, under View, select **Profile Summary Table** (or use the Profile Table icon as shown in the left margin.)

(e) From the Options menu, select **Include Profile Name in Table**.

(f) From the Options menu, select **Plans**, and then select the appropriate plans you want to compare to answer Question 1 below. In the same manner, you can select which **Profiles** and **Reaches** to display.

From the output, answer the following questions:

Question 1

For the **5-year** flood at R.K. **1.5** on the **Lower Kern River**, what is the water surface elevation for each plan?

	Plan	W.S. Elev (m)
1	Existing	
2	Levee_exQ	
3	ExGeom_dbQ	
4	Levee_dbQ	

Using the filled-in table above, what is the difference in the water surface elevations between the following plans?

Use a positive value to show that the w.s. elevation for the first plan is greater than the second. For example, if Plan 2 is 0.05 m higher than Plan 1, then enter +0.05 to show that the addition of the levee caused the water surface elevation to increase by 0.05 meters.

Plan 2 (Levee) vs. Plan 1 (Existing)? _____

Plan 3 (Detention Basin) vs. Plan 1 (Existing)? _____

Plan 4 (Levee + Detention Basin) vs. Plan 1 (Existing)? _____

Plan 3 (Detention Basin) vs. Plan 2 (Levee)? _____

Plan 4 (Levee + Detention Basin) vs. Plan 2 (Levee) _____

Plan 4 (Levee + Detention Basin) vs. Plan 3 (Detention Basin)? _____

Question 2

For the **10-year** flood at R.K. **1.5** on the **Lower Kern River**, why is the water surface elevation for Plan 2 (Levee) higher than for Plan 1 (Existing) even though both use the entire cross section for flow and have the same discharge?

Question 3

Would you expect what happened for question 2 to happen if we had used the normal ineffective flow area option and set the station and elevation of overtopping at the same point as the levee?

Why or why not?

Question 4

What is the backwater effect (in meters) of the proposed levee only project at R.K. **0.8** on **Cyrus Wash** for the **5-year** flood, i.e. Plan 2 (Levee) versus Plan 1 (Existing)?

Question 5

From the answer to question 4, what should we have done to the model to adequately measure the effects of the project?

Question 6

For the **5-year** flood (profile 2) on the **Lower Kern River** for Plan 2 (Levee), is the lateral extent of the water surface at R.K. **1.0** consistent with R.K. **1.5**?

Explain.

Question 7

Should we have used the same reach boundary conditions as under existing conditions when the levee was in place for Plan 2?

Congratulations! You've completed the Creating Plans HEC-RAS workshop.

WORKSHOP SOLUTIONS

WEST CONSULTANTS, INC.

HEC-RAS WORKSHOP NO. 3

CREATING PLANS

Question 1

For the **5-year** flood at R.K. **1.5** on the **Lower Kern River**, what is the water surface elevation for each plan?

	Plan Short ID	W.S. Elev (m)
1	Existing	4.59
2	Levee_exQ	4.61
3	ExGeom_dbQ	4.51
4	Levee_dbQ	4.66

Using the filled-in table above, what is the difference in the water surface elevations between the following plans?

Use a positive value to show that the w.s. elevation for the first plan is greater than the second. For example, if Plan 2 is 0.05 m higher than Plan 1, then enter +0.05 to show that the addition of the levee caused the water surface elevation to increase by 0.05 meters.

Plan 2 (Levee) vs. Plan 1 (Existing)? *+ 0.02 m*

Plan 3 (Detention Basin) vs. Plan 1 (Existing)? *– 0.08 m*

Plan 4 (Levee + Detention Basin) vs. Plan 1 (Existing)? *+ 0.07 m*

Plan 3 (Detention Basin) vs. Plan 2 (Levee)? *– 0.10 m*

Plan 4 (Levee + Detention Basin) vs. Plan 2 (Levee)? *+ 0.05 m*

Plan 4 (Levee + Detention Basin) vs. Plan 3 (Detention Basin)? *+ 0.15 m*

Question 2

For the **10-year** flood at R.K. **1.5** on the **Lower Kern River**, why is the water surface elevation for Plan 2 (Levee) higher than for Plan 1 (Existing) even though both use the entire cross section for flow and have the same discharge?

Plan 2 (Levee) = 5.26 m Plan 1 (Existing) = 5.24 m

This document is property of WEST Consultants, Inc., and is not to be duplicated without prior consent.

Plan 2 still has wetted perimeter of the levee even though it is overtopped [Cross Section Plot and Profile Output Table].

Question 3

Would you expect what happened for question 2 to happen if we had used the normal ineffective flow area option and set the station and elevation of overtopping at the same point as the levee? Why or why not?

No. If we had used the normal ineffective flow area option to represent the levee and the levee was overtopped, the water surface would have been the same as if the levee had never existed, which is the case of Plan 1 (Existing).

Question 4

What is the backwater effect (in meters) of the proposed levee only project at R.K. **0.8** on **Cyrus Wash** for the **5-year** flood in Plan 2 (Levee) versus Plan 1 (Existing)?

Plan 2 (Levee) = 5.16 m Plan 1 (Existing) = 5.07 m Increase of 0.09 m

[Profile Output Table]

Question 5

From the answer to question 4, what should we have done to the model to adequately measure the effects of the project?

We should have added more cross sections to extend the model upstream of cross section 0.8 on Cyrus Wash [Cross Section Output Table or Profile Output Table].

Question 6

For the **5-year** flood (profile 2) on the **Lower Kern River** for Plan 2 (Levee), is the lateral extent of the water surface at R.K. **1.0** consistent with R.K. **1.5**? Explain.

It is not consistent because the levee is overtopped at R.K. 1.5 but at R.K. 1.0, which is downstream of 1.5, all the water is contained within the levee. This means that somehow, the water went from the left overbank at R.K. 1.5 over the levee and into the channel, leaving the left overbank of R.K. 1.0 dry. These are not physically based results. We need to adjust the levee elevations to get consistent results even though the selected elevations may not be at actual levee elevations. The results show that the top elevation of the levee at R.K. 1.5 would need to be raised, given the remainder of the input data is correct.

Question 7

Should we have used the same reach boundary conditions as under existing conditions when the levee was in place for Plan 2?

This document is property of WEST Consultants, Inc., and is not to be duplicated without prior consent.

If proper techniques were used to set up the model initially, the answer would be "yes." The model should be started well downstream of the project area so that any errors or uncertainty in the boundary conditions will not affect the evaluation of results at the location of the project. If proper modeling techniques are followed, the same boundary conditions should apply for existing and project conditions. If the levees are reducing the area of flow at the downstream boundary, we would expect to have a different starting water surface elevation or a different estimate of the energy slope for the same discharge.

This document is property of WEST Consultants, Inc., and is not to be duplicated without prior consent.

WEST CONSULTANTS, INC.

HEC-RAS WORKSHOP NO. 7

TROUBLE SHOOTING

Objective: The goal of this workshop is to give you experience identifying and correcting stability issues in RAS unsteady flow models.

Time to Complete: 60 minutes

This workshop was originally developed through an extensive effort by the staff and engineers at the US Army Corps of Engineers Hydrologic Engineering Center (HEC), Davis, CA.

Intro

Introduction

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

Step 1

Stability Problem Number 1

The stream for this example is a section of Beaver Creek located near Kentwood, Louisiana. A project file (**Unsteady_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.

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.

Stepping through a time series animation with the Profile Plot is a good starting point to view where and when instabilities occurred.

When the Profile Plot 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, skewing 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. Then select **Options** and **Full Plot** and this should restore the view. You can then proceed animating through the time steps for analysis.

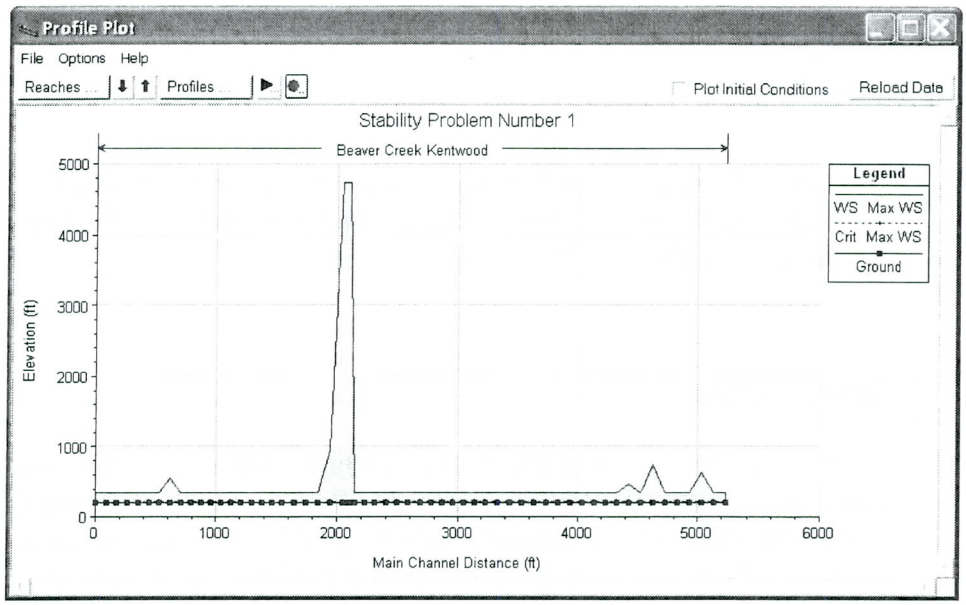


Figure 1. Maximum Water Surface Plot for a Simulation with Instabilities

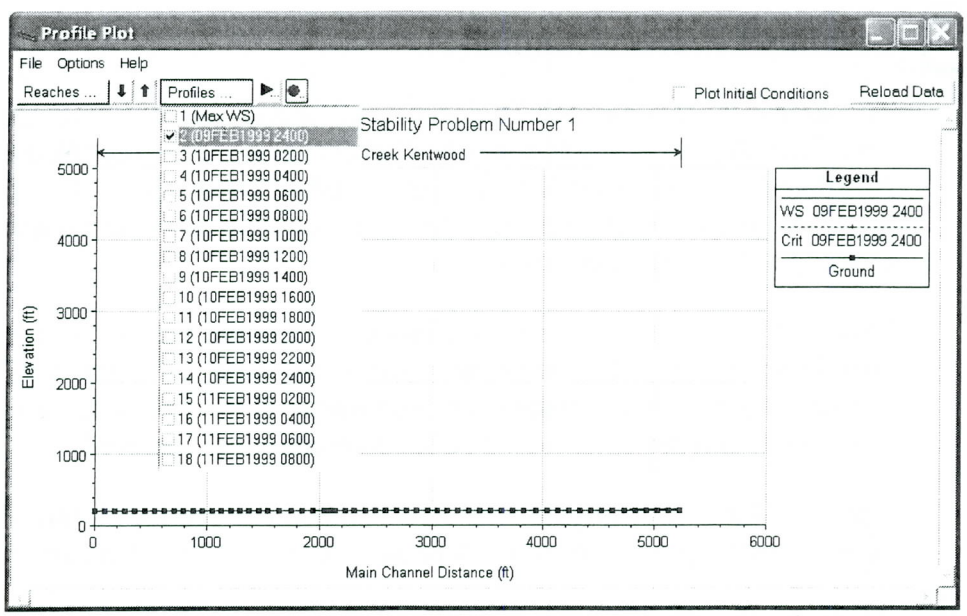


Figure 2. Initial Profile Selected

Question 1 At what time step and at what location did the program begin to go unstable?

Question 2 What caused the instability?

Question 3 If you were able to fix the instability, how was this achieved?

Step 2 Stability Problem Number 2

The stream for this example is a section of Beaver Creek located near Kentwood, Louisiana. A project file (**Unsteady_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, a lateral weir with gates, and a storage area.

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

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.

Question 1 At what time step and at what location did the program begin to go unstable?

Question 2 What caused the instability?

Question 3 If you were able to fix the instability, how was this achieved?

Step 3 Stability Problem Number 3

The stream for this example is a section of Beaver Creek located near Kentwood, Louisiana. A project file (**Unsteady_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 1 minute. Determine what is causing the instability <u>without adjusting this interval</u>.

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.

Question 1 At what time step and at what location did the program begin to go unstable?

Question 2 What caused the instability?

Question 3 If you were able to fix the instability, how was this achieved?

Step 4 **Stability Problem Number 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 (**Unsteady_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.

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.

Question 1 At what time step and at what location did the program begin to go unstable?

Question 2 What caused the instability?

Question 3 If you were able to fix the instability, how was this achieved?

WEST CONSULTANTS, INC.

HEC-RAS WORKSHOP NO. 7
TROUBLE SHOOTING

Stability Problem Number 1

Question 1

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

The HEC-RAS Hydraulic Computations window indicates that the first error occurred at RS 5.0 (Figure 1). Since this is near the downstream end of the reach the instability might be due to the boundary condition. This particular data set has a rating curve at the downstream boundary. Figure 2 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.

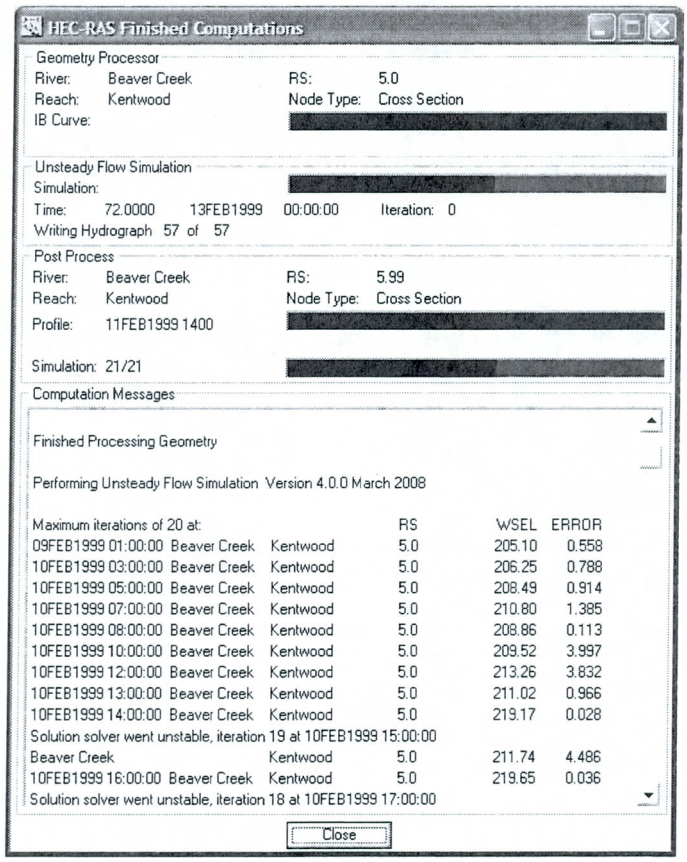


Figure 1

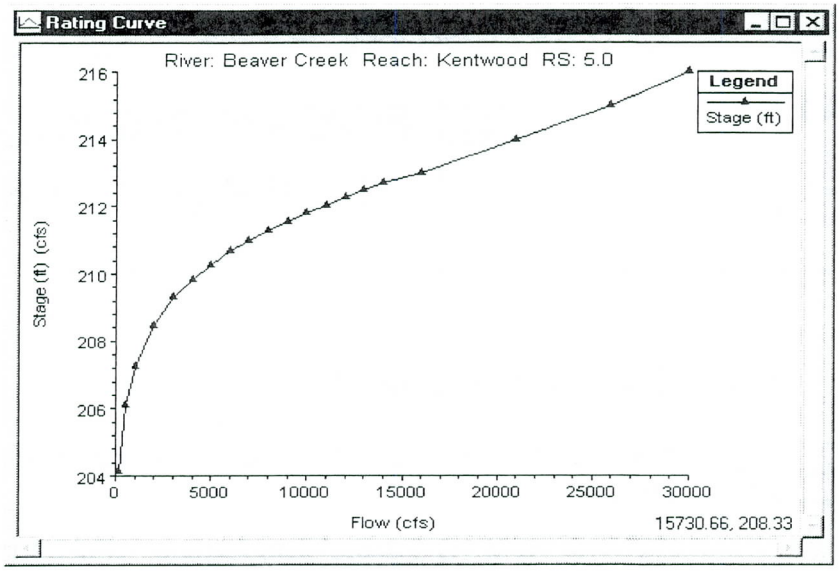


Figure 2. Downstream Boundary

Question 2

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 Editor, 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.

Question 3

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 3, the change produced a stable simulation.

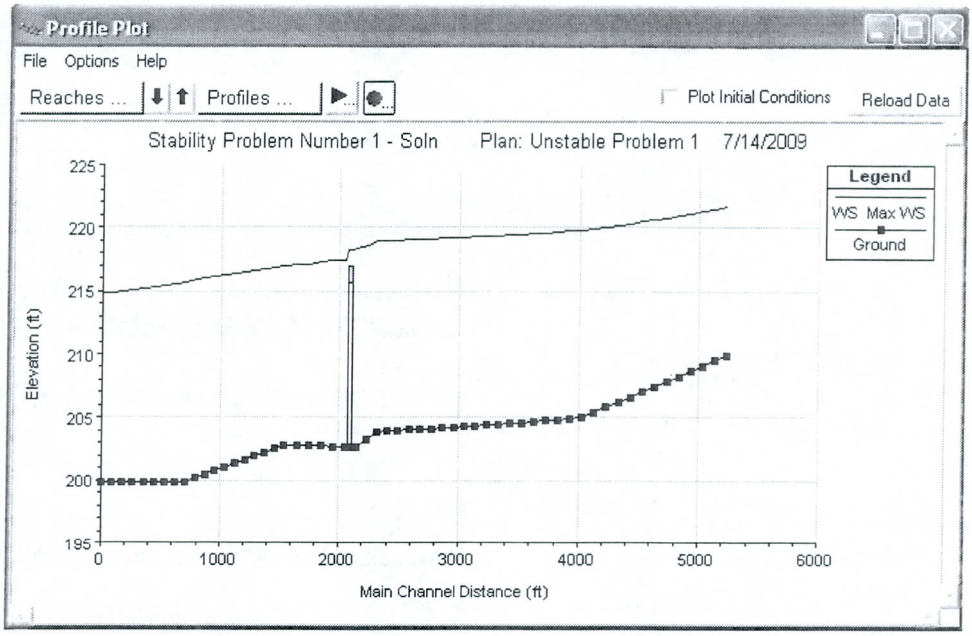
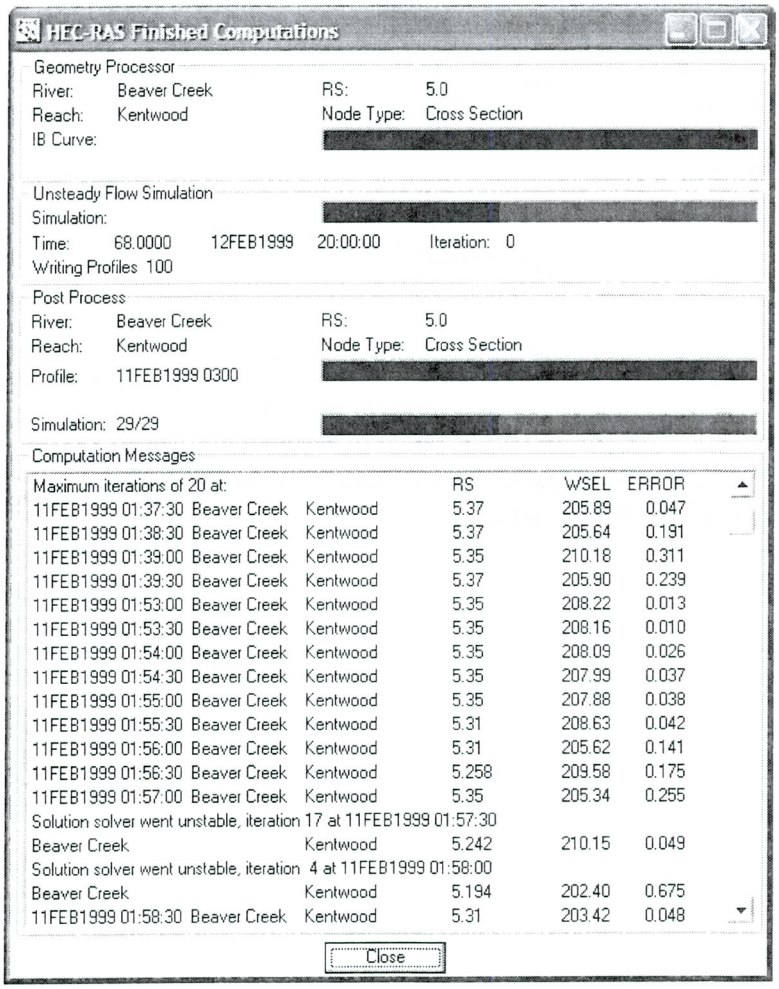


Figure 3. Maximum Water Surface Profile with a Computation Interval of 5 Minutes.

Stability Problem Number 2

The simulation was run and a number of errors are listed in the HEC-RAS Hydraulic Computations window (Figure 4).



HEC-RAS Finished Computations

Geometry Processor
River: Beaver Creek RS: 5.0
Reach: Kentwood Node Type: Cross Section
IB Curve:

Unsteady Flow Simulation
Simulation:
Time: 68.0000 12FEB1999 20:00:00 Iteration: 0
Writing Profiles 100

Post Process
River: Beaver Creek RS: 5.0
Reach: Kentwood Node Type: Cross Section
Profile: 11FEB1999 0300

Simulation: 29/29

Computation Messages

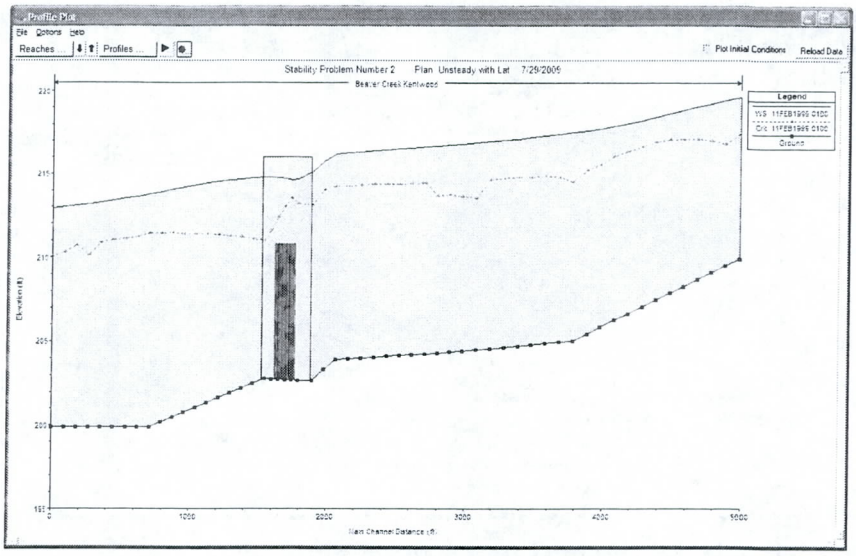
Maximum iterations of 20 at:			RS	WSEL	ERROR
11FEB1999 01:37:30	Beaver Creek	Kentwood	5.37	205.89	0.047
11FEB1999 01:38:30	Beaver Creek	Kentwood	5.37	205.64	0.191
11FEB1999 01:39:00	Beaver Creek	Kentwood	5.35	210.18	0.311
11FEB1999 01:39:30	Beaver Creek	Kentwood	5.37	205.90	0.239
11FEB1999 01:53:00	Beaver Creek	Kentwood	5.35	208.22	0.013
11FEB1999 01:53:30	Beaver Creek	Kentwood	5.35	208.16	0.010
11FEB1999 01:54:00	Beaver Creek	Kentwood	5.35	208.09	0.026
11FEB1999 01:54:30	Beaver Creek	Kentwood	5.35	207.99	0.037
11FEB1999 01:55:00	Beaver Creek	Kentwood	5.35	207.88	0.038
11FEB1999 01:55:30	Beaver Creek	Kentwood	5.31	208.63	0.042
11FEB1999 01:56:00	Beaver Creek	Kentwood	5.31	205.62	0.141
11FEB1999 01:56:30	Beaver Creek	Kentwood	5.258	209.58	0.175
11FEB1999 01:57:00	Beaver Creek	Kentwood	5.35	205.34	0.255
Solution solver went unstable, iteration 17 at 11FEB1999 01:57:30					
Beaver Creek		Kentwood	5.242	210.15	0.049
Solution solver went unstable, iteration 4 at 11FEB1999 01:58:00					
Beaver Creek		Kentwood	5.194	202.40	0.675
11FEB1999 01:58:30	Beaver Creek	Kentwood	5.31	203.42	0.048

Close

Figure 4

Question 1 *At what time step and at what location did the program begin to go unstable?*

The Profile Plot was examined and the animation tool was used to progress through the time steps. By stepping through the simulation, the animation shows that the program begins to go unstable between 11FEB1999 0100 and 11FEB1999 0200 (Figure 5). This corresponds with the first maximum iteration warning.



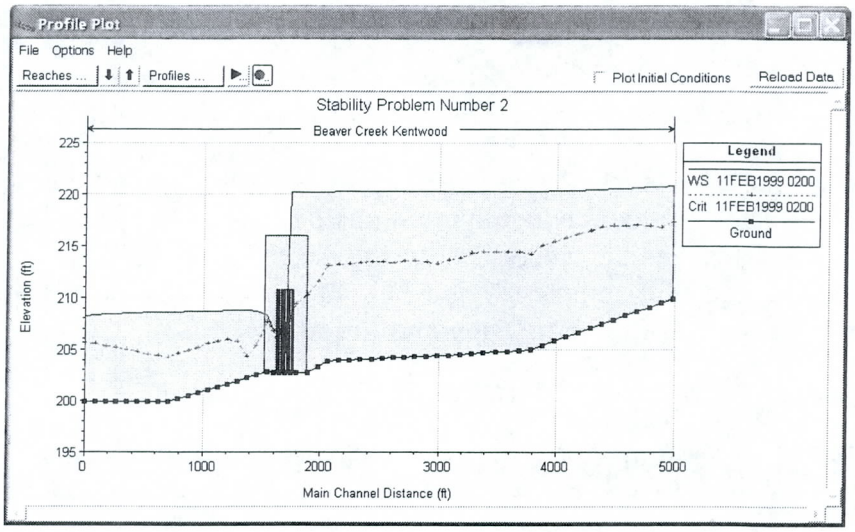


Figure 5

Question 2 *What caused the instability?*

The profile plot indicates the problem might be due to the downstream boundary condition as the instability first appears at the downstream end when stepping through the profile plot. However, a check of the downstream rating curve does not reveal any apparent problems. The abrupt fluctuation in stage at the downstream boundary might also indicate the problem is due to sudden changes in model. The **Time Series of Gate Openings** under the Unsteady Flow Data Editor was then examined. As can be seen on Figure 6, the gates are rapidly opened (1 minute interval).

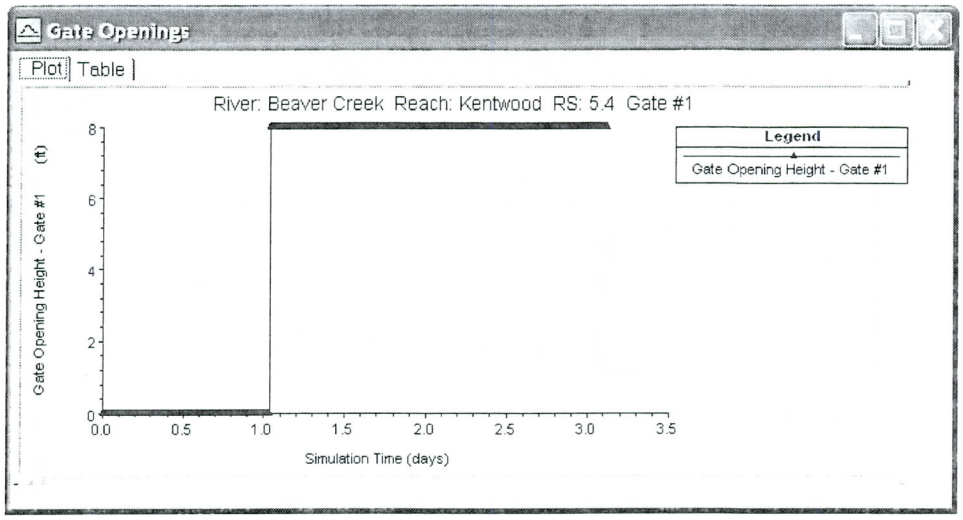


Figure 6. Plot of Gate Opening Time Series

Question 3

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 Editor. Values between the simulation time of 2300 and 2400 hours were deleted, and the values entered to have the gate open to 8 feet over 1 hour instead of 1 minute (Figure 7). The profile plot now displays realistic answers and the simulation runs to completion.

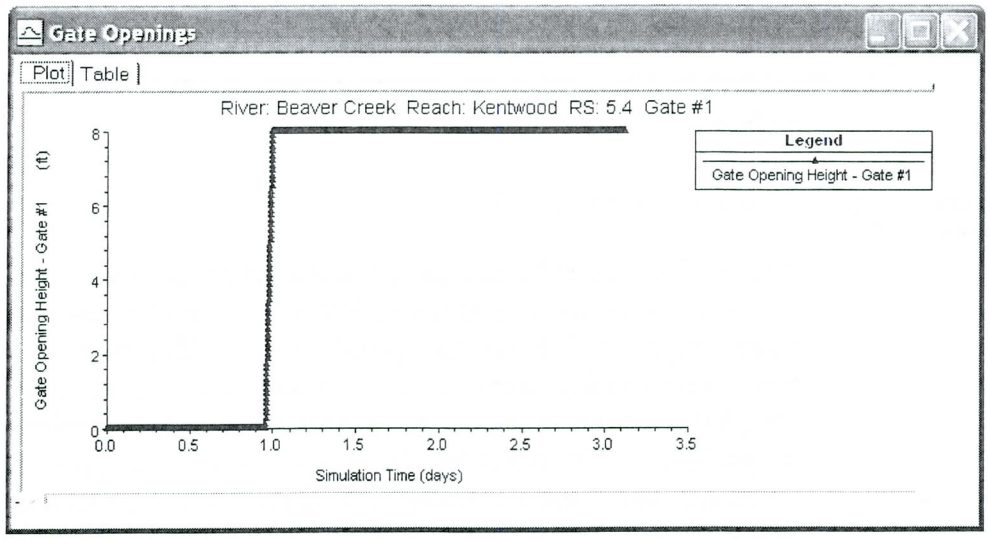


Figure 7

Using the slower gate opening hydrograph, the simulation still has a number of maximum iteration warnings, but the overall solution is stable and the results look reasonable. By opening the gates at a slower rate the rapid change in flow leaving the reach was decreased. This prevented instabilities in the simulation. The time step could also have been reduced, but you were asked not to do that for this problem.

Stability Problem Number 3

Examining the animation of the Profile Plot showed that the program crashed at the beginning of the simulation. The detailed output was examined to determine exactly when the instability occurred.

Question 1

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

The HEC-RAS Hydraulic Computations window shows the maximum iterations warning starting on 10FEB1999 at 0116 (Figure 8). These occur at the beginning of the simulation period and near the downstream end of the model. The model crashes on 10FEB1999 at 0249.

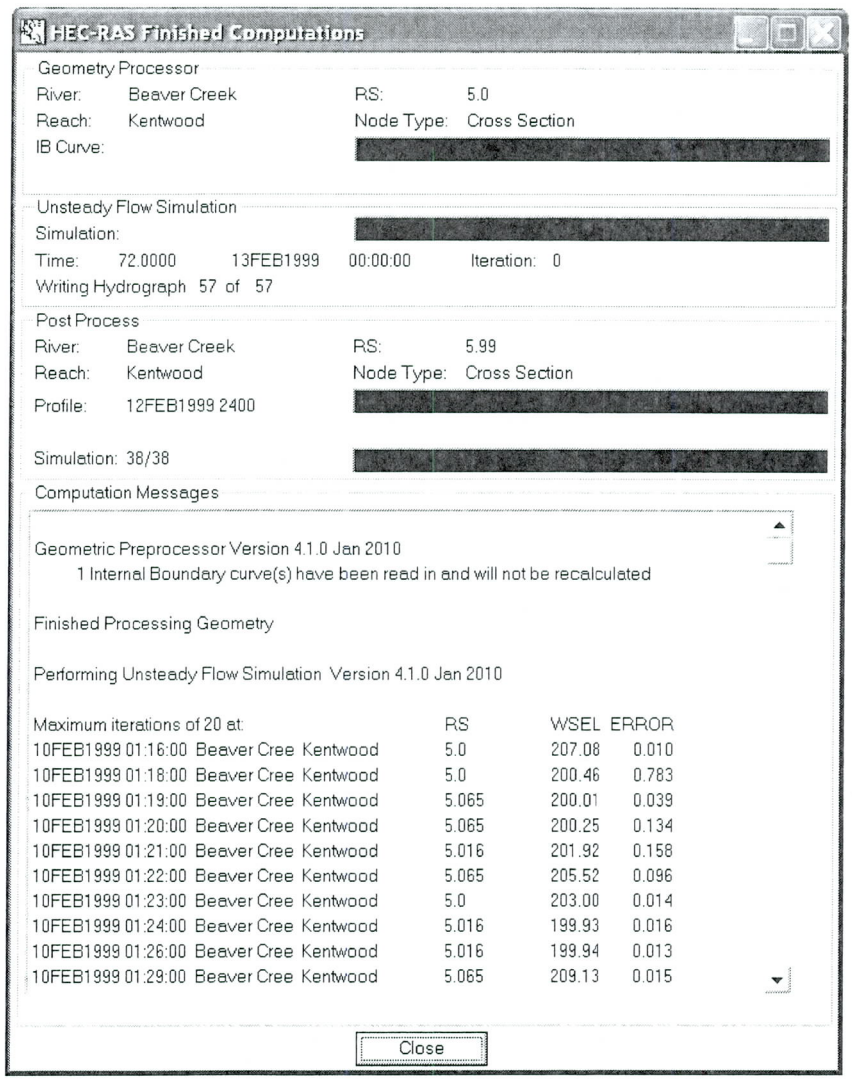


Figure 8. HEC-RAS Hydraulic Computations Window

Question 2 *What caused the instability?*

The profile plot was animated but having detailed output at 2 hour increments does not reveal the source of the error. The Hydrograph output Interval and Detailed Output Interval was changed to 5 minutes (and the simulation duration temporarily shorted to end at 2400 on 10FEB1999) and the profile plots reviewed. As shown in the progression of Figure 9, Figure 10, and Figure 11, this error begins at the downstream end of the reach.

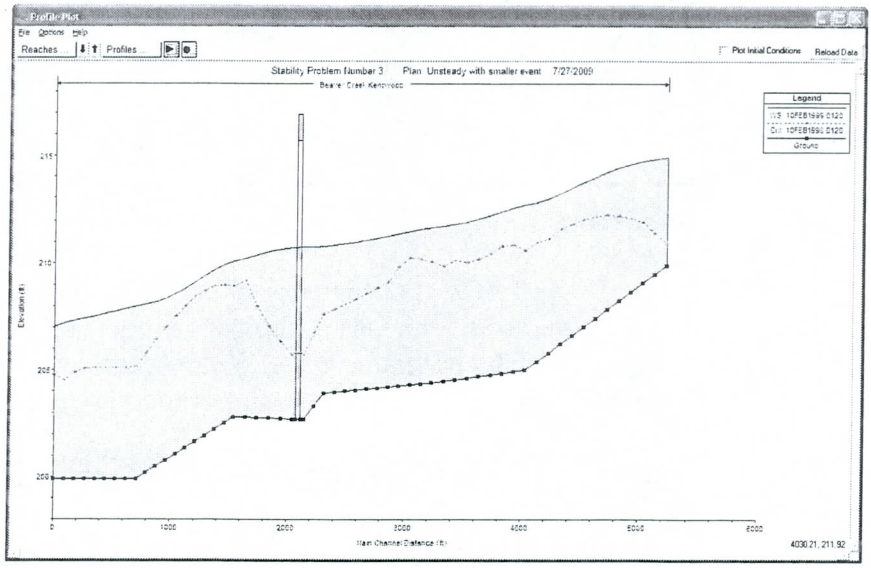


Figure 9

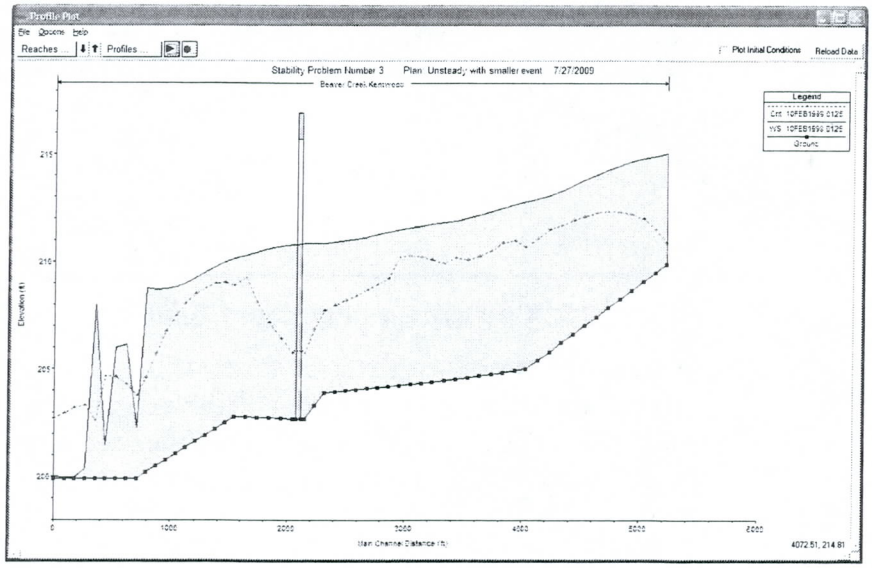


Figure 10

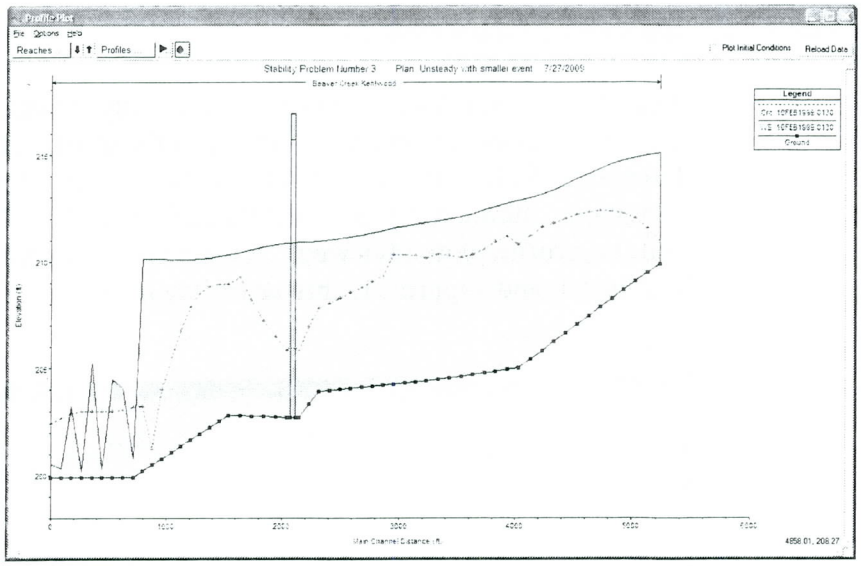


Figure 11

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.

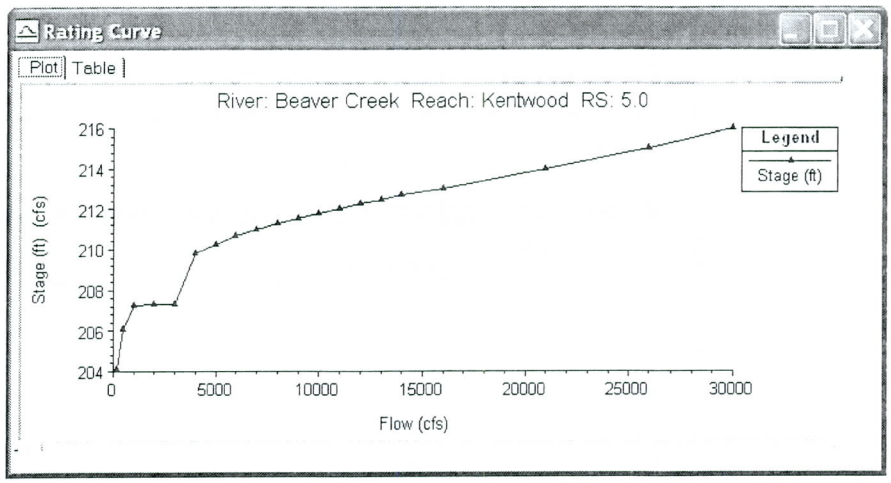


Figure 12. Downstream Boundary Condition

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.

Question 3 *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.

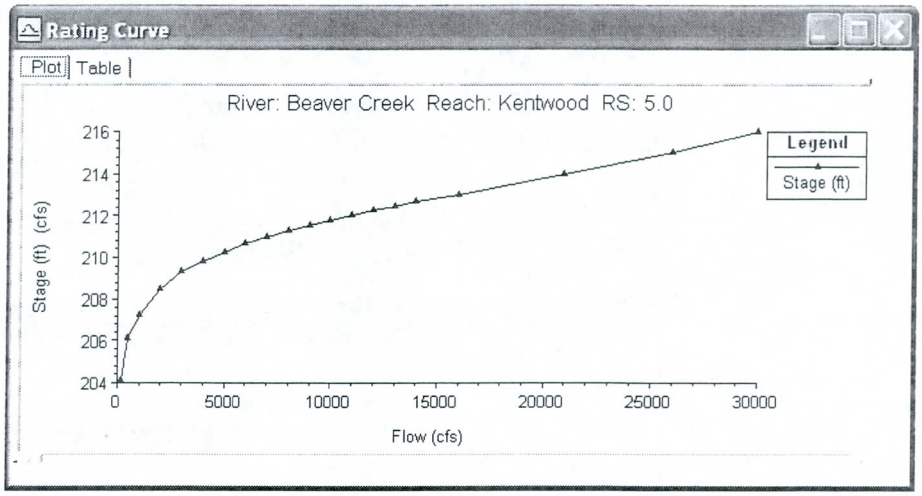


Figure 13. Adjusted Rating Curve for Stable Simulation

Stability Problem Number 4

The Profile Plot for this example shows a dramatic change in slope occurring near River Station 6.99.

Question 1

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

The HEC-RAS Hydraulic Computations window indicates there are a number of errors, the first of which start at RS 7.0 (Figure 14) near the beginning of the simulation time. Examining the animation of the Profile Plot shows that the water surface elevation and slope change dramatically over time near this location (Figure 15).

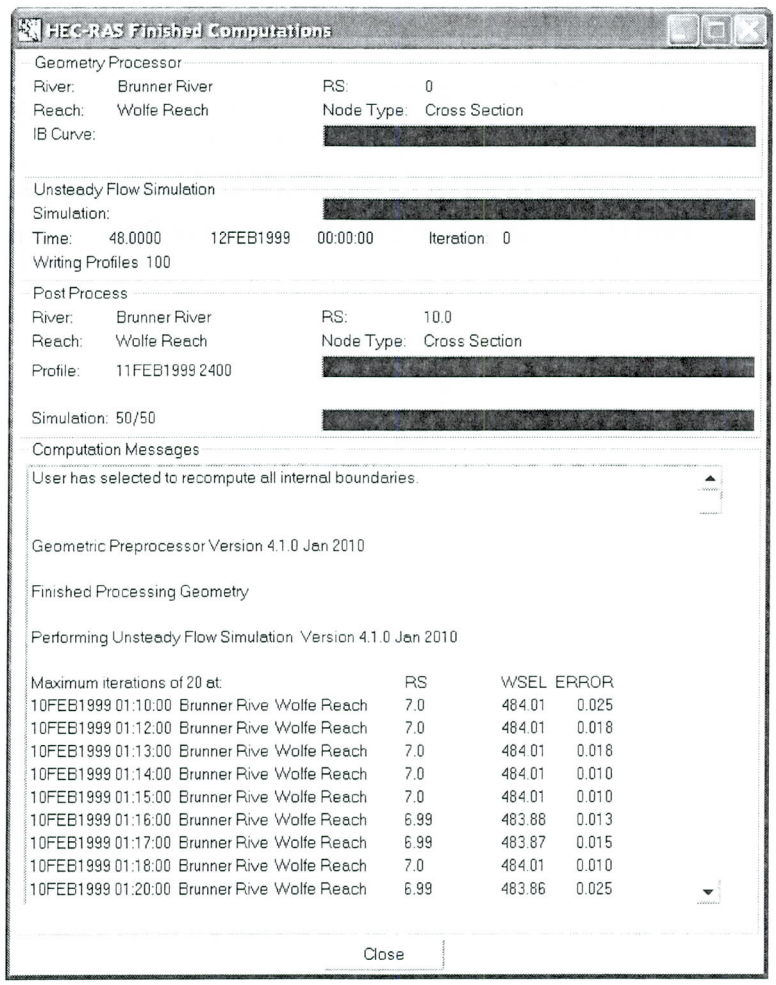


Figure 14

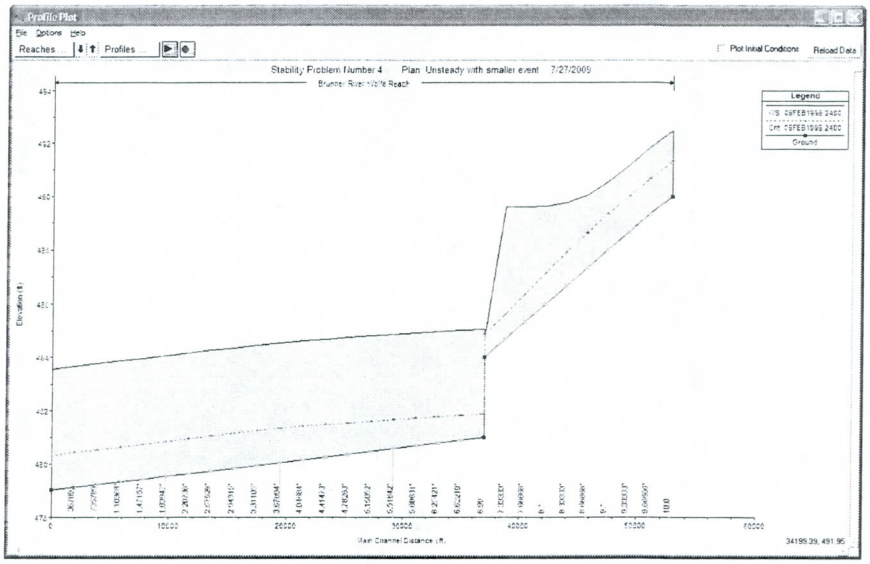


Figure 15. Profile Plot

Question 2 *What caused the instability?*

As seen on Figure 15, the water surface profile goes to critical depth where the slope of the reach changes. The current plan is not set to account for "mixed mode" (i.e., a combination of sub and super-critical flows).

Question 3 *If you were able to fix the instability, how was this achieved?*

Provided here are three possible solutions for this problem. The first involves examining the base flow from the input hydrograph. The simulation with the instability has a base flow and initial condition of 700 cfs (Figure 16). This low flow allows the water surface to pass through critical depth. One solution is to increase the base flow of 1000 cfs (Figure 17). The initial condition was also changed to 1000 cfs and the simulation re-run. This adjustment to the hydrograph and initial conditions proved a stable simulation.

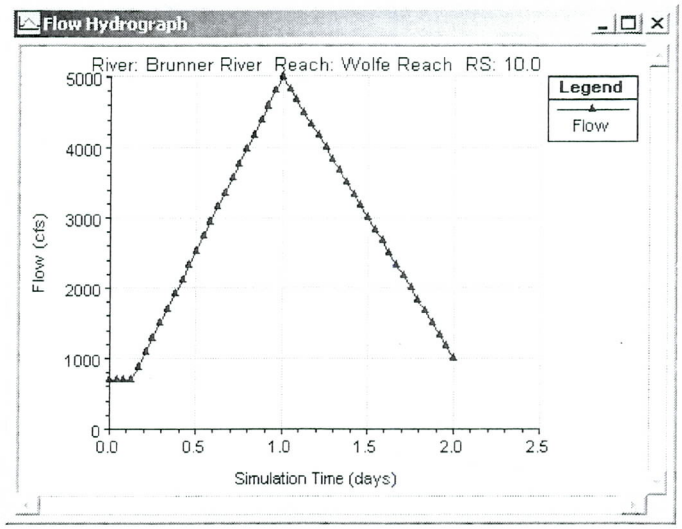


Figure 16

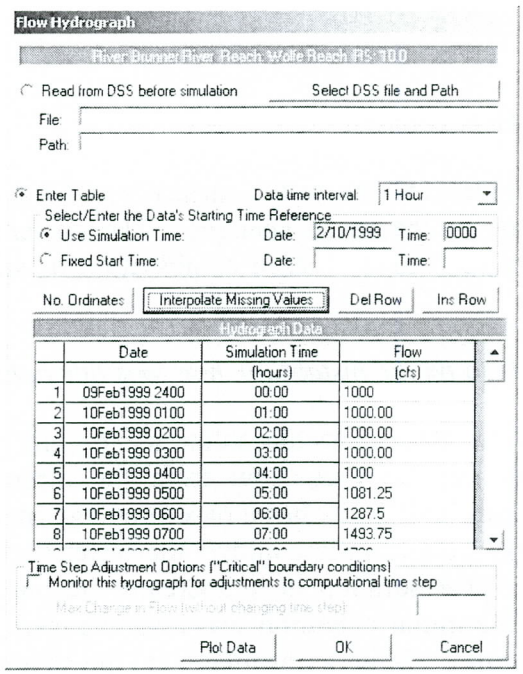


Figure 17

Additionally, the change in slope could be modeled as an inline structure. An inline structure was added just downstream of the change in slope at River Station 6.995. Figure 18 displays the cross section view of the inline structure. Figure 19 displays the dimensions of the inline structure.

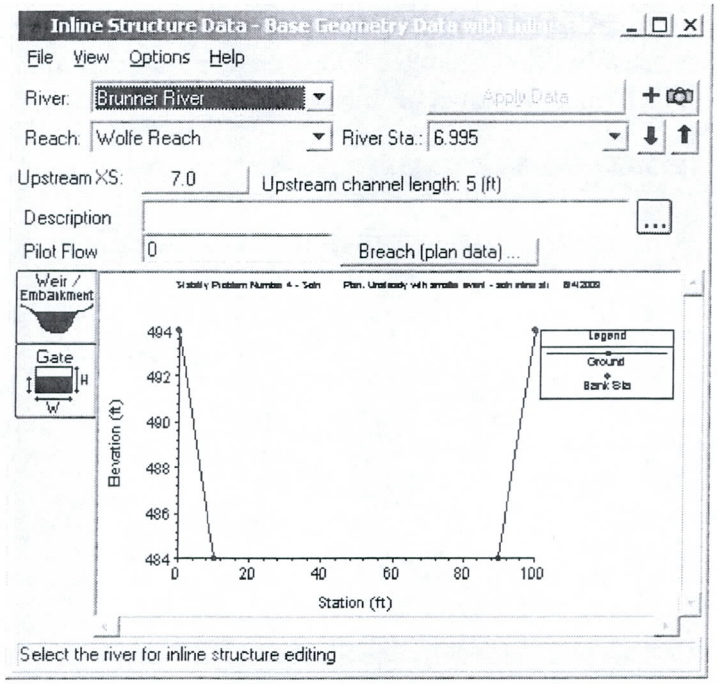


Figure 18. Plot of Inline Structure

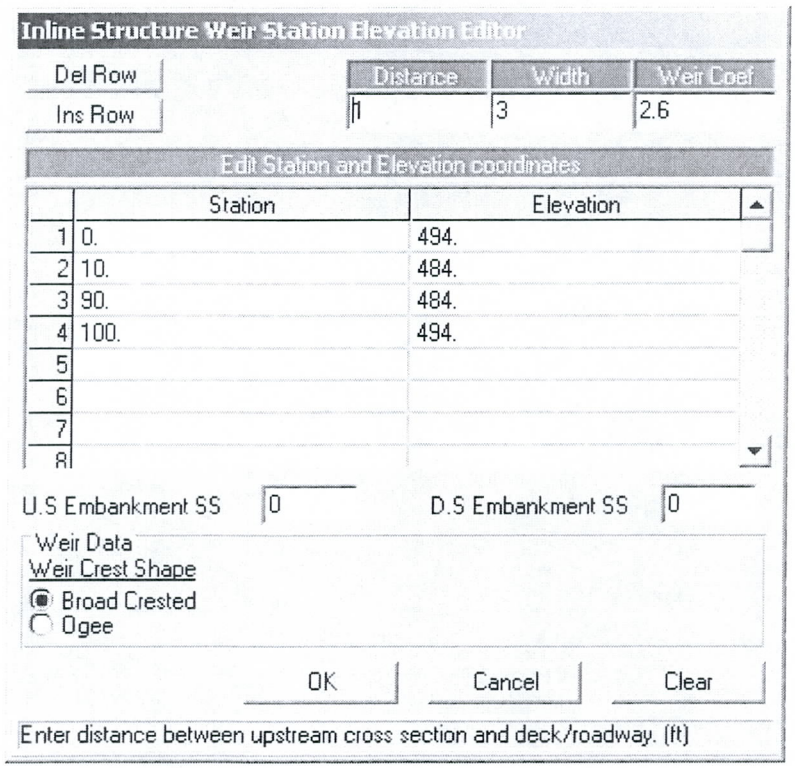


Figure 19. Dimensions of Inline Structure

A third option would be to use the **Mixed Flow Regime** feature in the Unsteady Flow Analysis Editor. A comparison of the three solutions at 1100 hours is shown in Figure 20.

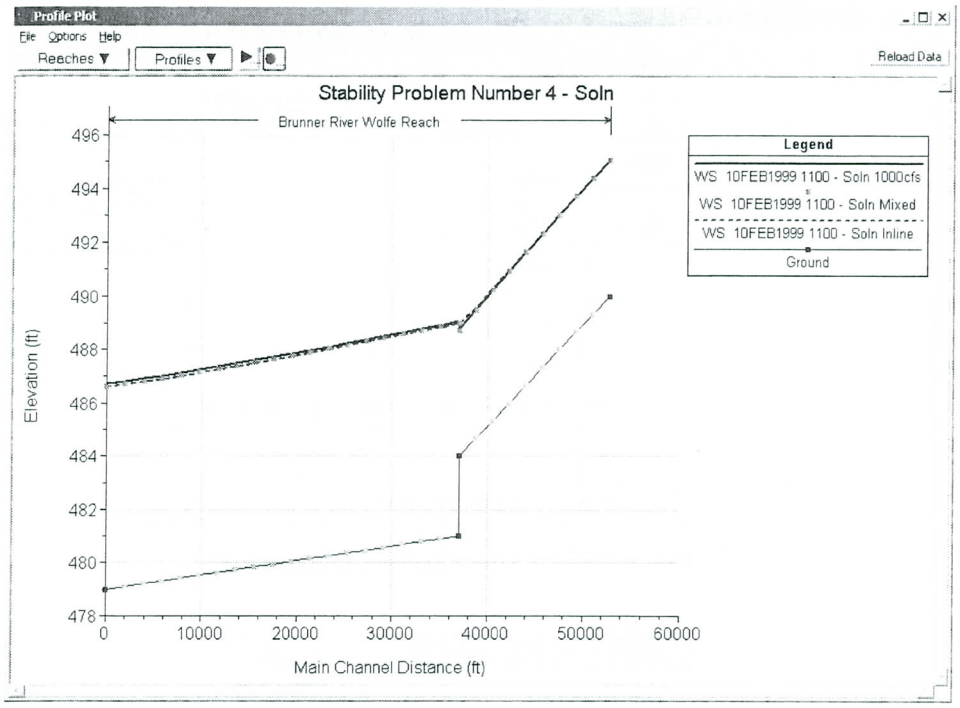


Figure 20