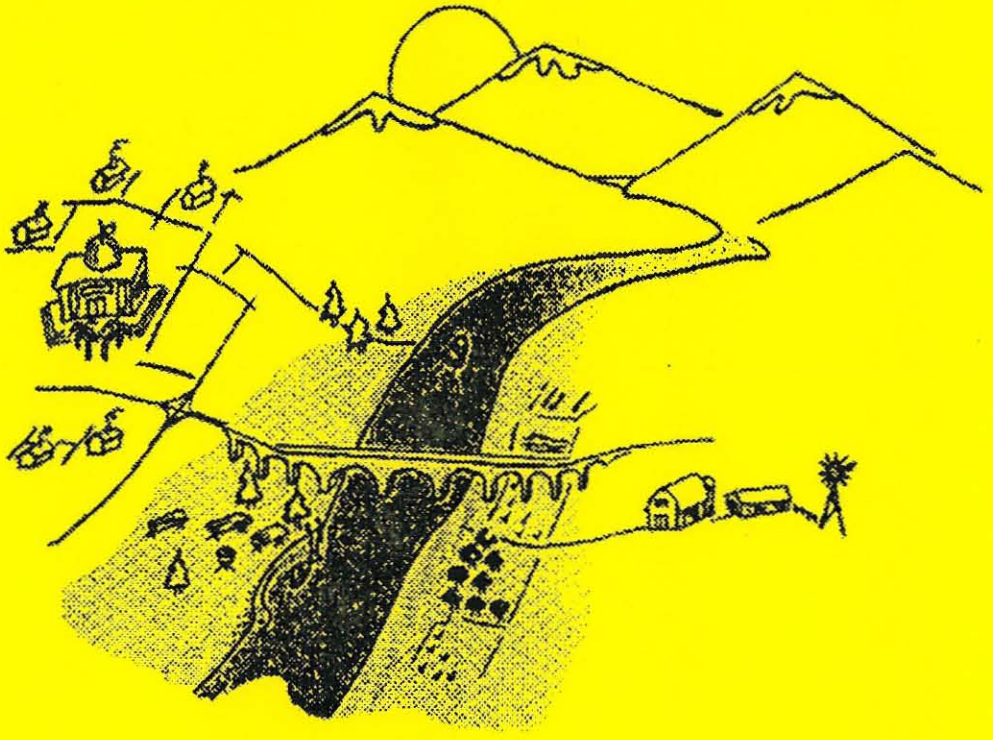
US Army Corps
of Engineers
Hydrologic Engineering Center

HEC-RAS
River Analysis System



User's Manual

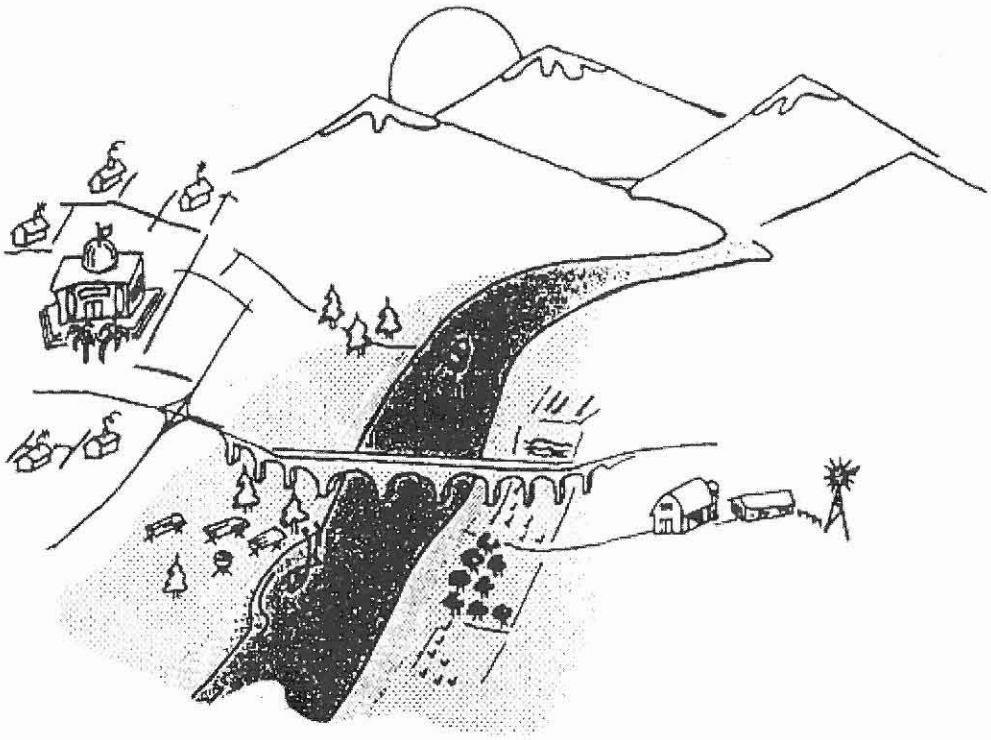
Version 3.1
November 2002

Approved for Public Release. Distribution Unlimited

CPD-68

US Army Corps
of Engineers
Hydrologic Engineering Center

HEC-RAS
River Analysis System



User's Manual

Version 3.1
November 2002

Approved for Public Release. Distribution Unlimited

CPD-68

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the date needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY *(Leave blank)*	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	November 2002	Computer Program Documentation

4. TITLE AND SUBTITLE
HEC-RAS, River Analysis System User's Manual

5. FUNDING NUMBERS

6. AUTHOR(S)
Gary W. Brunner

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
US ARMY CORPS OF ENGINEERS
HYDROLOGIC ENGINEERING CENTER (HEC)
609 Second Street
Davis, CA 95616-4687

8. PERFORMING ORGANIZATION REPORT NUMBER
CPD-68

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)

10. SPONSORING / MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION / AVAILABILITY STATEMENT
Distribution is unlimited.

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

The U.S. Army Corps of Engineers' River Analysis System (HEC-RAS) is software that allows you to perform one-dimensional steady and unsteady flow river hydraulics calculations.

HEC-RAS is an integrated system of software, designed for interactive use in a multi-tasking, multi-user network environment. The system is comprised of a graphical user interface (GUI), separate hydraulic analysis components, data storage and management capabilities, graphics and reporting facilities.

The HEC-RAS system will ultimately contain three one-dimensional hydraulic analysis components for: (1) steady flow water surface profile computations; (2) unsteady flow simulation; and (3) movable boundary sediment transport computations. A key element is that all three components will use a common geometric data representation and common geometric and hydraulic computation routines. In addition to the three hydraulic analysis components, the system contains several hydraulic design features that can be invoked once the basic water surface profiles are computed.

The current version of HEC-RAS supports Steady and Unsteady flow water surface profile calculations. New features and additional capabilities will be added in future releases.

14. SUBJECT TERMS
water surface profiles, river hydraulics, steady and unsteady flow, computer program

15. NUMBER OF PAGES
420

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED	UNLIMITED

HEC-RAS

River Analysis System

User's Manual

Version 3.1

November 2002

US Army Corps of Engineers
Institute For Water Resources
Hydrologic Engineering Center
609 Second Street
Davis, CA 95616

(530) 756-1104
(530) 756-8250 FAX
www.hec.usace.army.mil

River Analysis System, HEC-RAS
Software Distribution and Availability Statement

The HEC-RAS executable code and documentation are public domain software that were developed by the Hydrologic Engineering Center for the U.S. Army Corps of Engineers. The software was developed at the expense of the United States Federal Government, and is therefore in the public domain. This software can be downloaded for free from our internet site (www.hec.usace.army.mil). HEC cannot provide technical support for this software to non-Corps users. See our software vendor list (on our web page) to locate organizations that provide the program, documentation, and support services for a fee. However, we will respond to all documented instances of program errors. Documented errors are bugs in the software due to programming mistakes not model problems due to user-entered data.

Cover sketch adapted from:

Flood Plain Management Program, Handbook for Public Officials
Department of Water Resources
State of California
August 1986

HEC-RAS User's Manual
Table of Contents

Foreword viii

Chapter 1 Introduction 1-1
General Philosophy of the Modeling System 1-2
Overview of Program Capabilities 1-2
User Interface 1-2
Hydraulic Analysis Components 1-3
Data Storage and Management 1-4
Graphics and Reporting 1-4
HEC-RAS Documentation 1-5
Overview of This Manual 1-5

Chapter 2 Installing HEC-RAS 2-1
Hardware and Software Requirements 2-2
Installation Procedure 2-2
Uninstall Procedure 2-3

Chapter 3 Working With HEC-RAS - An Overview 3-1
Starting HEC-RAS 3-2
Steps in Developing a Hydraulic Model With HEC-RAS 3-6
Starting a New Project 3-6
Entering Geometric Data 3-7
Entering Flow Data and Boundary Conditions 3-9
Performing The Hydraulic Computations 3-10
Viewing and Printing Results 3-12
Importing HEC-2 Data 3-16
What You Should Know First 3-16
Steps For Importing HEC-2 Data 3-19
Reproducing HEC-2 Results 3-20
Getting and Using Help 3-22

Chapter 4 Example Application 4-1
Starting a New Project 4-2
Entering Geometric Data 4-3
Drawing the Schematic of the River System 4-3
Entering Cross Section Data 4-4
Entering Junction Data 4-9
Saving the Geometry Data 4-10
Entering Steady Flow Data 4-10
Performing the Hydraulic Calculations 4-13
Viewing Results 4-14
Printing Graphics and Tables 4-20
Sending Graphics Directly to the Printer 4-20
Sending Graphics to the Windows Clipboard 4-20
Sending Tables Directly to the Printer 4-21
Sending Tables to the Windows Clipboard 4-21

Exiting the Program 4-22

Chapter 5 Working With Projects 5-1
Understanding Projects 5-1
Elements of a Project 5-2
Plan Files 5-2
Run Files 5-2
Output Files 5-3
Geometry Files 5-3
Steady Flow Data Files 5-3
Unsteady Flow Data Files 5-3
Sediment Data Files 5-4
Hydraulic Design Data Files 5-4
Creating, Opening, Saving, Renaming, and Deleting Projects 5-6
Project Options 5-6

Chapter 6 Entering and Editing Geometric Data 6-1
Developing the River System Schematic 6-2
Building the Schematic 6-2
Adding Tributaries into an Existing Reach 6-4
Editing the Schematic 6-4
Interacting With the Schematic 6-7
Background Pictures 6-9
Cross Section Data 6-10
Entering Cross Section Data 6-10
Editing Cross Section Data 6-12
Cross Section Options 6-13
Plotting Cross Section Data 6-21
Stream Junctions 6-21
Entering Junction Data 6-21
Selecting a Modeling Approach 6-22
Bridges and Culverts 6-23
Cross Section Locations 6-24
Contraction and Expansion Losses 6-26
Bridge Hydraulic Computations 6-26
Entering and Editing Bridge Data 6-28
Bridge Design Editor 6-42
Culvert Hydraulic Computations 6-44
Entering and Editing Culvert Data 6-45
Bridge and Culvert Options 6-50
Bridge and Culvert View Features 6-52
Multiple Bridge and/or Culvert Openings 6-53
Entering Multiple Opening Data 6-54
Defining The Openings 6-57
Multiple Opening Calculations 6-58
Inline Structures (Weirs and Gated Spillways) 6-59
Entering and Editing Inline Structure Data 6-59
Lateral Structures (Weirs, Gated Spillways, Culverts, and Diversion Rating Curves) 6-67
Entering and Editing Lateral Structure Data 6-67

Linear Routing Option 6-77
Storage Areas 6-79
Storage Area Connections 6-81
Pump Stations 6-85
Cross Section Interpolation 6-89
River Ice 6-95
Entering and Editing Ice Data 6-95
Entering Ice Data at a Cross Section 6-95
Entering Ice Data Through a Table 6-98
Entering Ice Data at Bridges 6-99
Setting Tolerances for the Ice Jam Calculations 6-100
Viewing and Editing Data Through Tables 6-101
Manning's n or k values 6-101
Reach Lengths 6-103
Contraction and Expansion Coefficients 6-103
Bank Stations 6-104
Levees 6-104
Ice Cover 6-104
River Stationing 6-105
Node Names 6-105
Node Descriptions 6-106
Picture File Associations 6-106
Bridge Width Table 6-106
Weir and Gate Coefficient Table 6-107
Importing Geometric Data 6-107
GIS Format 6-108
USACE Survey Data Format 6-108
HEC-2 Data Format 6-108
HEC-RAS Data Format 6-108
UNET Geometric Data Format 6-109
MIKE 11 Cross Section Data 6-109
Geometric Data Tools 6-109
Graphical Cross Section Editor 6-109
Reverse Stationing Data 6-111
Set Ineffective Areas to Permanent Mode 6-111
Cross Section Points Filter 6-112
Fixed Sediment Elevations 6-114
Pilot Channels 6-115
GIS Cut Line Check 6-117
View Reach Connectivity 6-118
Datum Adjustment 6-118
Fix Overlapping Ineffective Areas 6-118
GIS Coordinates 6-118
Plot GIS Bounds 6-119
Attaching and Viewing Pictures 6-119
Saving the Geometric Data 6-121

Chapter 7 Performing a Steady Flow Analysis 7-1
Entering and Editing Steady Flow Data 7-1
Steady Flow Data 7-1

Boundary Conditions 7-2
Steady Flow Data Options 7-4
Saving the Steady Flow Data 7-7
Importing Data from the HEC Data Storage System (HEC-DSS) 7-7
Performing Steady Flow Calculations 7-10
Defining a Plan 7-11
Saving the Plan Information 7-11
Simulation Options 7-11
Starting the Computations 7-17

Chapter 8 Performing an Unsteady Flow Analysis 8-1
Entering and Editing Unsteady Flow Data 8-1
Unsteady Flow Data 8-1
Boundary Conditions 8-2
Initial Conditions 8-9
Unsteady Flow Data Options 8-11
Saving The Unsteady Flow Data 8-12
Performing Unsteady Flow Calculations 8-13
Defining A Plan 8-13
Selecting Programs to Run 8-14
Simulation Time Window 8-18
Computation Settings 8-19
Simulation Options 8-20
Saving The Plan Information 8-31
Starting The Computations 8-32
Calibration of Unsteady Flow Models 8-33
Observed Hydrologic Data 8-33
River and Floodplain Geometry 8-35
Roughness Coefficients 8-36
River and Floodplain Storage 8-41
Hydraulic Structure Coefficients 8-43
Steps To Follow In The Calibration Process 8-44
General Trends When Adjusting Model Parameters 8-44
Calibration Suggestions and Warning's 8-46
Model Accuracy, Stability, and Sensitivity 8-47
Model Accuracy 8-47
Model Stability 8-47
Model Sensitivity 8-60

Chapter 9 Viewing Results 9-1
Cross Sections, Profiles, and Rating Curves 9-1
Viewing Graphics on the Screen 9-1
Graphical Plot Options 9-4
Plotting Velocity Distribution Output 9-6
Plotting Other Variables in Profile 9-8
Plotting One Variable Versus Another 9-8
Sending Graphics to the Printer or Plotter 9-9
Sending Graphics to the Windows Clipboard 9-10
Stage and Flow Hydrographs 9-10
X-Y-Z Perspective Plots 9-11

Tabular Output 9-13
Detailed Output Tables 9-13
Detailed Output Table Options 9-17
Profile Summary Tables 9-17
User Defined Output Tables 9-20
Sending Tables to the Printer 9-21
Sending Tables to the Windows Clipboard 9-22
Viewing Results From the River System Schematic 9-22
Viewing Ice Information 9-24
Viewing Graphical Ice Information on the Screen 9-24
Viewing Tabular Ice Information 9-26
Viewing Data Contained in an HEC-DSS File 9-26
Exporting Results To HEC-DSS 9-30

Chapter 10 Performing a Floodway Encroachment Analysis ... 10-1
General 10-2
Entering Floodplain Encroachment Data 10-3
Performing the Floodplain Encroachment Analysis 10-6
Viewing the Floodplain Encroachment Results 10-7
Floodway Encroachments With Unsteady Flow 10-11

Chapter 11 Troubleshooting With HEC-RAS 11-1
Built in Data Checking 11-1
Checking The Data as it is Entered 11-1
Checking Data Before Computations are Performed 11-2
Errors, Warnings, and Notes 11-3
Log Output 11-6
Steady Flow Log Output 11-6
Unsteady Flow Log Output 11-7
Viewing The Log File 11-7
Reviewing and Debugging the Normal Output 11-8
Viewing Graphics 11-8
Viewing Tabular Output 11-8
The Occurrence of Critical Depth 11-8
Computational Program Does Not Run To Completion 11-9

Chapter 12 Computing Scour at Bridges 12-1
General Modeling Guidelines 12-2
Entering Bridge Scour Data 12-3
Entering Contraction Scour Data 12-4
Entering Pier Scour Data 12-6
Entering Abutment Scour Data 12-10
Computing Total Bridge Scour 12-13

Chapter 13 Performing Channel Modifications 13-1
General Modeling Guidelines 13-2
Entering Channel Modification Data 13-2
Performing the Channel Modifications 13-6
Comparing Existing and Modified Conditions 13-8

Chapter 14 Using GIS Data With HEC-RAS 14-1
General Modeling Guidelines 14-2
Importing GIS/CADD Data Into HEC-RAS 14-4
Completing The Data and Performing The Computations 14-7
Completing The Geometric Data 14-7
Entering Additional Cross Section Data 14-7
Performing The Computations and Viewing Results 14-9
Exporting Computed Results to The GIS or CADD 14-10

Chapter 15 Stable Channel Design Functions 15-1
General Modeling Guidelines 15-2
General Command Buttons 15-2
Uniform Flow Computations 15-2
Solving For Slope, Discharge, and W/S Elevation 15-2
Solving For Bottom Width 15-4
Applying Uniform Flow Data To The Geometry File 15-7
Saving Uniform Flow Data 15-7
Stable Channel Design 15-8
Copeland Method 15-8
Regime Method 15-12
Tractive Force Method 15-13
Sediment Transport Capacity 15-15

Chapter 16 Advanced Features For Unsteady Flow Routing 16-1
Mixed Flow Regime 16-2
Dam Break Analysis 16-6
Entering Dam Break Data 16-7
Estimating Dam Break Parameters 16-11
HEC-RAS Output For Dam Break Analyses 16-12
Levee Overtopping and Breaching 16-14
Modeling Pump Stations 16-21
Navigation Dams 16-25
Pool Only Control 16-26
Hinge Point Only Control 16-28
Hinge Point and Minimum Pool Operations 16-31
Hinge Point and Minimum and Maximum Pool Control 16-34

Appendix A References A-1

Appendix B HEC-RAS Import/Export Files For Geospatial Data B-1

Appendix C List of HEC-RAS Output Variables C-1

CHAPTER 1

Introduction

Welcome to the U.S. Army Corps of Engineers River Analysis System (HEC-RAS) developed by the Hydrologic Engineering Center. This software allows you to perform one-dimensional steady flow, unsteady flow calculations. Sediment transport computations will be added in a future version.

The HEC-RAS modeling system was developed as a part of the Hydrologic Engineering Center's "Next Generation" (NexGen) of hydrologic engineering software. The NexGen project encompasses several aspects of hydrologic engineering, including: rainfall-runoff analysis; river hydraulics; reservoir system simulation; flood damage analysis; and real-time river forecasting for reservoir operations.

This chapter discusses the general philosophy of HEC-RAS and gives a brief overview of the capabilities of the modeling system. Documentation for HEC-RAS is discussed, as well as an overview of this manual.

Contents

- General Philosophy of the Modeling System
- Overview of Program Capabilities
- HEC-RAS Documentation
- Overview of This Manual

General Philosophy of the Modeling System

HEC-RAS is an integrated system of software, designed for interactive use in a multi-tasking environment. The system is comprised of a graphical user interface (GUI), separate hydraulic analysis components, data storage and management capabilities, graphics and reporting facilities.

The HEC-RAS system will ultimately contain three one-dimensional hydraulic analysis components for: (1) steady flow water surface profile computations; (2) unsteady flow simulation; and (3) movable boundary sediment transport computations. A key element is that all three components will use a common geometric data representation and common geometric and hydraulic computation routines. In addition to the three hydraulic analysis components, the system contains several hydraulic design features that can be invoked once the basic water surface profiles are computed.

The current version of HEC-RAS supports Steady and Unsteady flow water surface profile calculations. New features and additional capabilities will be added in future releases.

Overview of Program Capabilities

HEC-RAS is designed to perform one-dimensional hydraulic calculations for a full network of natural and constructed channels. The following is a description of the major capabilities of HEC-RAS.

User Interface

The user interacts with HEC-RAS through a graphical user interface (GUI). The main focus in the design of the interface was to make it easy to use the software, while still maintaining a high level of efficiency for the user. The interface provides for the following functions:

- File management
- Data entry and editing
- Hydraulic analyses
- Tabulation and graphical displays of input and output data
- Reporting facilities
- On-line help

Hydraulic Analysis Components

Steady Flow Water Surface Profiles. This component of the modeling system is intended for calculating water surface profiles for steady gradually varied flow. The system can handle a full network of channels, a dendritic system, or a single river reach. The steady flow component is capable of modeling subcritical, supercritical, and mixed flow regime water surface profiles.

The basic computational procedure is based on the solution of the one-dimensional energy equation. Energy losses are evaluated by friction (Manning's equation) and contraction/expansion (coefficient multiplied by the change in velocity head). The momentum equation is utilized in situations where the water surface profile is rapidly varied. These situations include mixed flow regime calculations (i.e., hydraulic jumps), hydraulics of bridges, and evaluating profiles at river confluences (stream junctions).

The effects of various obstructions such as bridges, culverts, weirs, and structures in the flood plain may be considered in the computations. The steady flow system is designed for application in flood plain management and flood insurance studies to evaluate floodway encroachments. Also, capabilities are available for assessing the change in water surface profiles due to channel improvements, and levees.

Special features of the steady flow component include: multiple plan analyses; multiple profile computations; multiple bridge and/or culvert opening analysis; and split flow optimization.

Unsteady Flow Simulation. This component of the HEC-RAS modeling system is capable of simulating one-dimensional unsteady flow through a full network of open channels. The unsteady flow equation solver was adapted from Dr. Robert L. Barkau's UNET model (Barkau, 1992 and HEC, 1997). This unsteady flow component was developed primarily for subcritical flow regime calculations. However, with the release of Version 3.1, the model can now performed mixed flow regime (subcritical, supercritical, hydraulic jumps, and draw downs) calculations in the unsteady flow computations module.

The hydraulic calculations for cross-sections, bridges, culverts, and other hydraulic structures that were developed for the steady flow component were incorporated into the unsteady flow module.

Sediment Transport/Movable Boundary Computations. This component of the modeling system is intended for the simulation of one-dimensional sediment transport/movable boundary calculations resulting from scour and deposition over moderate time periods (typically years, although applications to single flood events are possible).

The sediment transport potential is computed by grain size fraction, thereby allowing the simulation of hydraulic sorting and armoring. Major features will include the ability to model a full network of streams, channel dredging,

various levee and encroachment alternatives, and the use of several different equations for the computation of sediment transport.

The model will be designed to simulate long-term trends of scour and deposition in a stream channel that might result from modifying the frequency and duration of the water discharge and stage, or modifying the channel geometry. This system can be used to evaluate deposition in reservoirs, design channel contractions required to maintain navigation depths, predict the influence of dredging on the rate of deposition, estimate maximum possible scour during large flood events, and evaluate sedimentation in fixed channels.

Data Storage and Management

Data storage is accomplished through the use of "flat" files (ASCII and binary), as well as the HEC-DSS. User input data are stored in flat files under separate categories of project, plan, geometry, steady flow, unsteady flow, and sediment data. Output data is predominantly stored in separate binary files. Data can be transferred between HEC-RAS and other programs by utilizing the HEC-DSS.

Data management is accomplished through the user interface. The modeler is requested to enter a single filename for the project being developed. Once the project filename is entered, all other files are automatically created and named by the interface as needed. The interface provides for renaming, moving, and deletion of files on a project-by-project basis.

Graphics and Reporting

Graphics include X-Y plots of the river system schematic, cross-sections, profiles, rating curves, hydrographs, and many other hydraulic variables. A three-dimensional plot of multiple cross-sections is also provided. Tabular output is available. Users can select from pre-defined tables or develop their own customized tables. All graphical and tabular output can be displayed on the screen, sent directly to a printer (or plotter), or passed through the Windows Clipboard to other software, such as a word-processor or spreadsheet.

Reporting facilities allow for printed output of input data as well as output data. Reports can be customized as to the amount and type of information desired.

HEC-RAS Documentation

The HEC-RAS package includes several documents. Each document is designed to help the modeler learn to use a particular aspect of the modeling system. The documentation has been broken up into the following three categories:

Documentation	Description
User's Manual	This manual is a guide to using HEC-RAS. The manual provides an introduction and overview of the modeling system, installation instructions, how to get started, a simple example, detailed descriptions of each of the major modeling components, and how to view graphical and tabular output.
Hydraulic Reference Manual	This manual describes the theory and data requirements for the hydraulic calculations performed by HEC-RAS. Equations are presented along with the assumptions used in their derivation. Discussions are provided on how to estimate model parameters, as well as guidelines on various modeling approaches.
Applications Guide	This document contains a series of examples that demonstrate various aspects of HEC-RAS. Each example consists of a problem statement, data requirements, general outline of solution steps, displays of key input and output screens, and discussions of important modeling aspects.

Overview of This Manual

This user's manual is the primary piece of documentation on how to use the HEC-RAS system. The manual is organized as follows:

- Chapters 1-2 provide an introduction and overview of HEC-RAS, as well as instructions on how to install the software.

- Chapters 3-5 describe how to use the HEC-RAS software in a step-by-step procedure, including a sample problem that the user can follow along with. Understanding how this system works with projects is also discussed.

- Chapters 6-8 explain in detail how to enter and edit data, and how to perform the different types of analyses that are available.
- Chapter 9 provides detailed discussions on how to view graphical and tabular output, as well as how to develop user-defined tables.
- Chapter 10 describes how to perform a floodway encroachment analysis.
- Chapter 11 provides discussions on "Trouble Shooting" and understanding the most common Errors, Warnings, and Notes.
- Chapter 12 describes how to perform bridge scour computations from within HEC-RAS.
- Chapter 13 describes how to perform channel modifications within HEC-RAS.
- Chapter 14 explains how to utilize GIS/CADD data in HEC-RAS, as well as how to export HEC-RAS results back to the GIS/CADD system.
- Chapter 15 describes how to use the Hydraulic Design Functions in HEC-RAS for performing stable channel deign and analysis, as well as sediment transport potential calculations, and rip rap sizing.
- Chapter 16 contains topics on advanced uses of the unsteady flow simulation capabilities, including: mixed flow regime for unsteady flow; dam break analysis; levee overtopping and breaching; modeling pump stations; and navigation dams.
- Appendix A contains a list of references.
- Appendix B contains a detailed description of the file formats used for importing and exporting GIS data to and from HEC-RAS.
- Appendix C contains a description of all the output variables available from the HEC-RAS program.

CHAPTER 2

Installing HEC-RAS

You install HEC-RAS using the program SETUP.EXE. The Setup program installs the software, sample applications, and the Help system.

This chapter discusses the hardware and system requirements needed to use HEC-RAS, how to install the software, and how to uninstall the software.

Contents

- Hardware and Software Requirements
- Installation Procedure
- Uninstall Procedure

Important

You cannot simply copy files from the distribution CD to your hard disk and run HEC-RAS. You must use the Setup program, which decompresses and installs the files to the appropriate directories.

Hardware and Software Requirements

Before you install the HEC-RAS software, make sure that your computer has at least the minimum required hardware and software. In order to get the maximum performance from the HEC-RAS software, recommended hardware and software is shown in parentheses. This version of HEC-RAS will run on a microcomputer that has the following:

- Intel Based PC or compatible machine with Pentium processor or higher (a Pentium III or higher is recommended).
- A hard disk with at least 40 megabytes of **free** space (100 megabytes or more is recommended).
- A CD Rom drive (or CD-R, CD-RW, DVD).
- A minimum of 32 megabytes of RAM if using Windows 95, 98, ME or 64 megabytes if using Windows NT, 2000 or XP (128 or more is recommended).
- A mouse.
- Color Video Display (Recommend running in Super VGA (1024x768) or higher, and as large a monitor as possible).
- MS Windows 95, 98, ME, NT 4.0, 2000, or XP (or later versions).

Installation Procedure

Installation of the HEC-RAS software is accomplished through the use of the Setup program.

< **To install the software onto your hard disk do the following:**

1. Insert the HEC-RAS CD into your CD drive.
2. The setup program should run automatically.
3. If the setup program does not run, use the windows explorer to start the **setup.exe** program on the CD.
4. Follow the setup instructions on the screen.

The setup program automatically creates a program group called **HEC**. This program group will be listed under the **Programs** menu, which is under the **Start** menu. The HEC-RAS program icon will be contained within the HEC program group. The user can create a shortcut icon by opening Windows

Explorer and dragging the HEC-RAS executable onto the desktop. The HEC-RAS executable can be found in the c:\Program Files\hec\ras directory with the name "RAS.EXE".

Uninstall Procedure

The HEC-RAS Setup program automatically registers the software with the Windows operating system. To unistall the software, do the following:

- From the **Start Menu** select **Settings** and then **Control Panel**.
- From within the Control Panel folder select **Add/Remove Programs**.
- From the Tab marked as **Install/Uninstall** select the HEC-RAS program from the list of installed software, then press the **Add/Remove** button.
- Follow the uninstall directions on the screen and the software will be removed from your hard disk. It is up to the user to remove the HEC-RAS icon from the desktop.

CHAPTER 3

Working With HEC-RAS - An Overview

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 hydraulic design computations in the future.

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, sediment 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 the HEC-RAS, the modeler can easily formulate new plans. After simulations are made for the various plans, the results can be compared simultaneously in both tabular and graphical form.

This chapter provides an overview of how a study is performed with the HEC-RAS software. Special topics on how to import HEC-2 data, reproducing HEC-2 results, and how to use on-line help are also covered.

Contents

- Starting HEC-RAS
- Steps in Developing a Hydraulic Model With HEC-RAS
- Importing HEC-2 Data
- Reproducing HEC-2 Results
- Getting and Using Help

Starting HEC-RAS

When you run the HEC-RAS Setup program, you automatically get a new program group called **HEC** and program icon called **HEC-RAS**. They should appear in the start menu under the section called **Programs**. The user also has the option of creating a shortcut on the desktop. If a shortcut is created, the icon for HEC-RAS will look like the following:

HEC-RAS 3.1

Figure 3.1 The HEC-RAS Icon in Windows

To Start HEC-RAS from Windows:

- Double-click on the HEC-RAS Icon. If you do not have an HEC-RAS shortcut on the desktop, go to the **Start** menu and select **Programs**, then select **HEC**, and then **HEC-RAS**.

When you first start HEC-RAS, you will see the main window as shown in Figure 3.2 (except you will not have any project files listed on your main window).

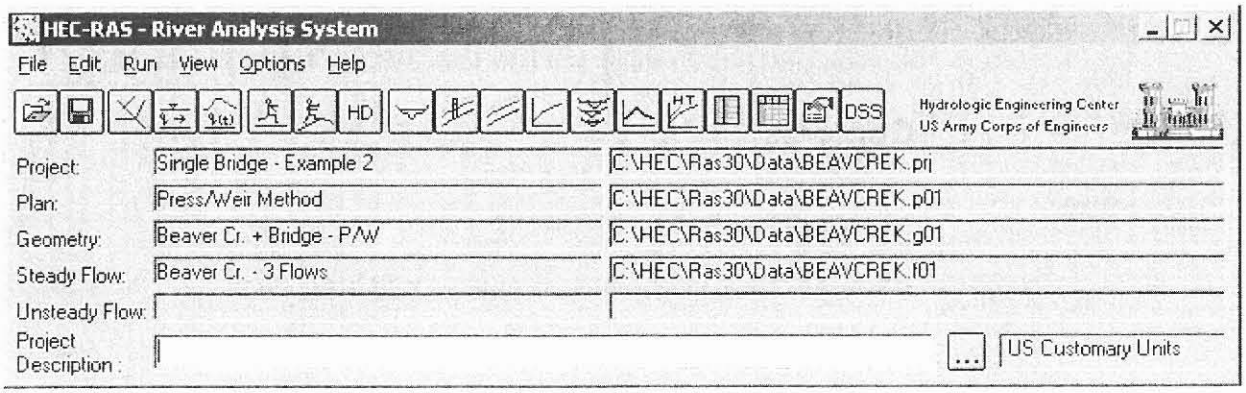


Figure 3.2 The HEC-RAS Main Window

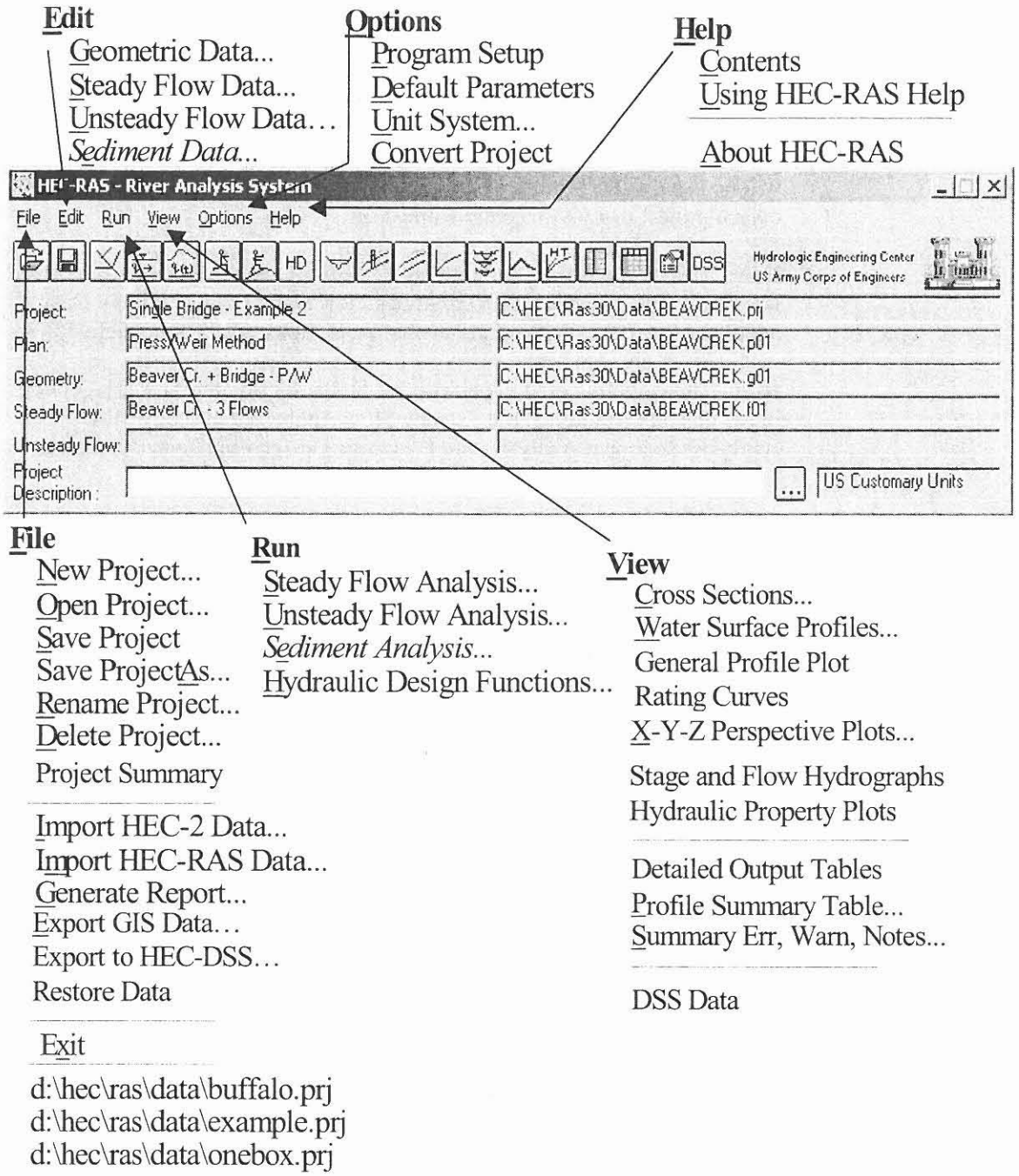


Figure 3.3 HEC-RAS Main Window Menu Bar Structure

At the top of the HEC-RAS main window is a Menu bar (Figure 3.3) with the following options:

File: This option is used for file management. Options available under the File menu include: New Project; Open Project; Save Project; Save Project As; Rename Project: Delete Project; Project Summary; Import HEC-2 Data; Import HEC-RAS data; Generate Report; Export GIS Data; Export to HEC-DSS; Restore Data; and Exit. In addition, the most recently opened projects will be listed at the bottom of the File menu, which allows the user to quickly open a project that was recently worked on.

Edit: This option is used for entering and editing data. Data are categorized into four types: Geometric Data; Steady Flow Data; Unsteady Flow Data; and Sediment Data. In the current version, Sediment Data is not active.

Run: This option is used to perform the hydraulic calculations. The options under this menu item include: Steady Flow Analysis; Unsteady Flow Analysis; Sediment Analysis; and Hydraulic Design Functions. In the current version, Sediment Analysis is not available.

View: This option contains a set of tools that provide for graphical and tabular displays of the model output. The View menu item currently includes: Cross Sections; Water Surface Profiles; General Profile Plot; Rating Curves; X-Y-Z Perspective Plots; Stage and Flow Hydrographs; Hydraulic Properties Plots; Detailed Output Tables; Profile Summary Tables; and Summary Err, Warn, Notes.

Options: This menu item allows the user to change Program Setup options; set Default Parameters; establish the Default Units System (U.S. Customary or Metric); and Convert Project Units (U.S. Customary to Metric, or Metric to U.S. Customary).

Help: This option allows the user to get on-line help, as well as display the current version information about HEC-RAS.

Also on the HEC-RAS main window is a Button bar (Figure 3.4). The Button bar provides quick access to the most frequently used options under the HEC-RAS menu bar. A description of each button is shown in Figure 3.4.

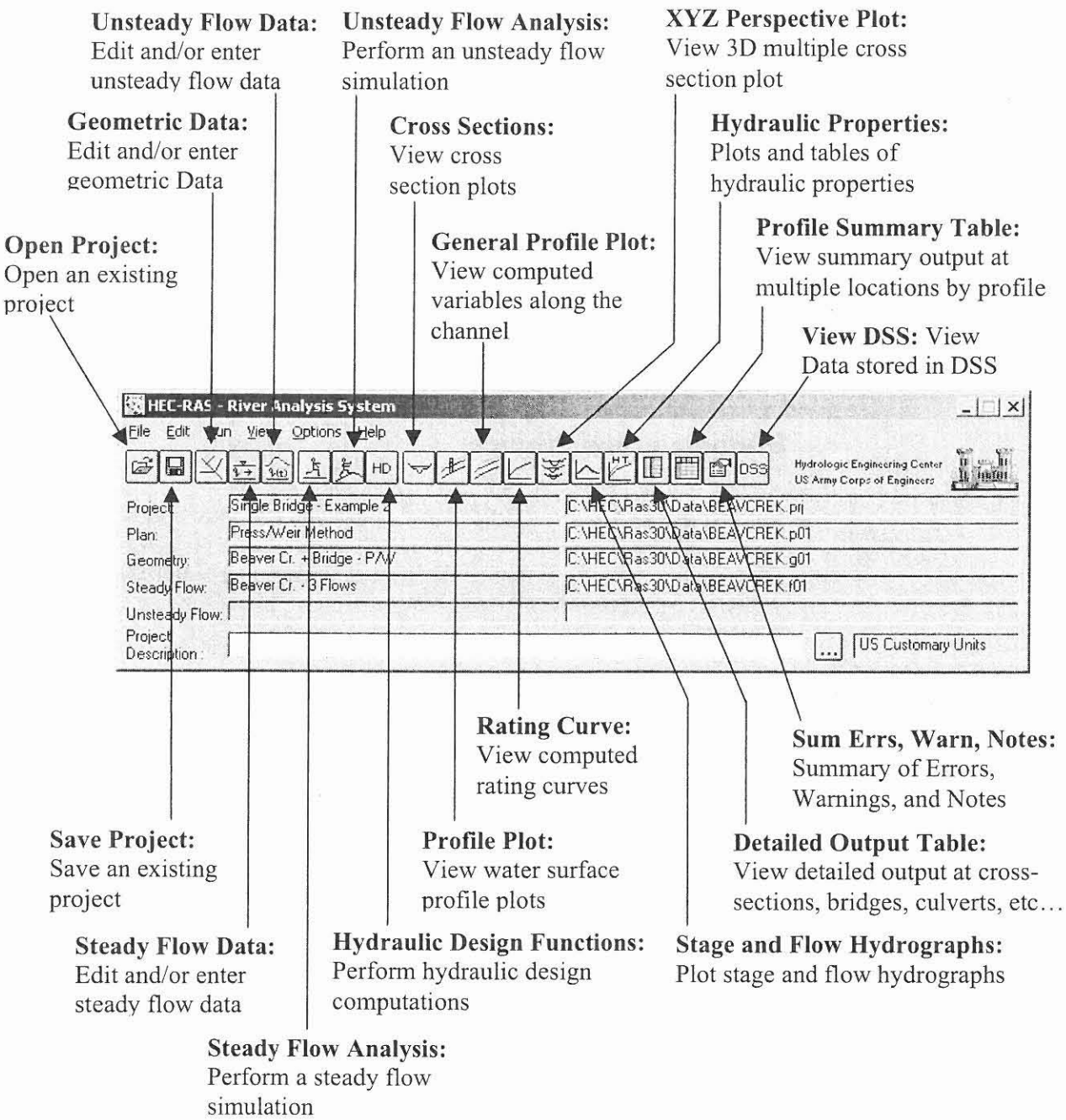


Figure 3.4 HEC-RAS Main Window Button Bar

Steps in Developing a Hydraulic Model with HEC-RAS

There are five main steps in creating a hydraulic model with HEC-RAS:

- Starting a new project
- Entering geometric data
- Entering flow data and boundary conditions
- Performing the hydraulic calculations
- Viewing and printing results

Starting a New Project

The first step in developing a hydraulic model with HEC-RAS is to establish which directory you wish to work in and to enter a title for the new project. To start a new project, go to the **File** menu on the main HEC-RAS window and select **New Project**. This will bring up a New Project window as shown in Figure 3.5.

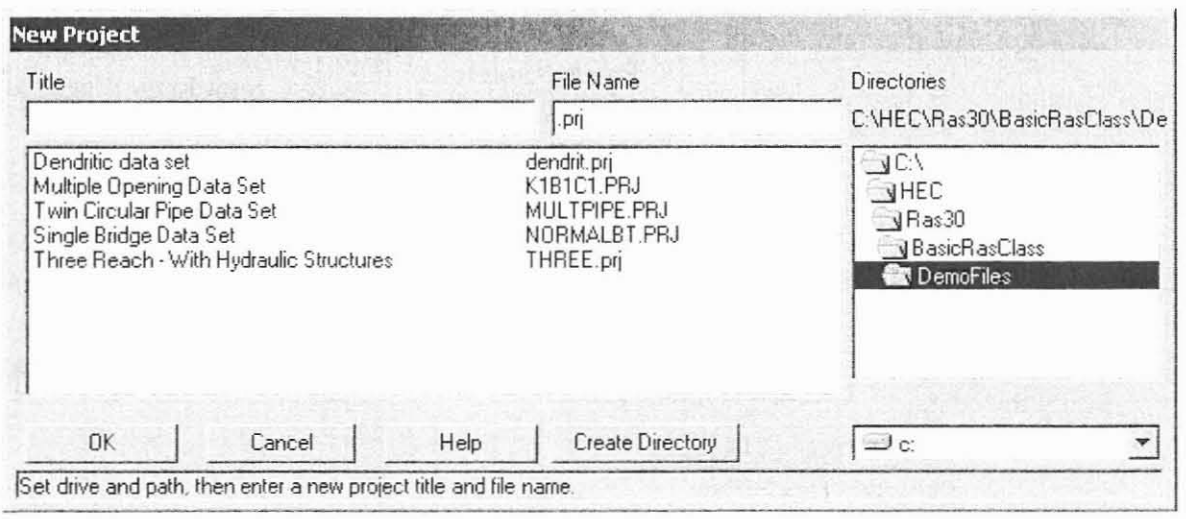


Figure 3.5 New Project Window

As shown in Figure 3.5, you first select the drive and path that they want to work in (to actually select a path you must double click the directory you want in the directory box), next enter a project title and file name. The project filename must have the extension ".prj", the user is not allowed to change this. Once you have entered all the information, press the "**OK**" button to have the information accepted. After the OK button is pressed, a message box will appear with the title of the project and the directory that the project is going to be placed in. If this information is correct, press the **OK**

button. If the information is not correct, press the **Cancel** button and you will be placed back into the **New Project** window.

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 step is accomplished by selecting **Unit System** from the **Options** menu of the main HEC-RAS window.

Entering Geometric Data

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 Figure 3.6 (except yours will be blank when you first bring this screen up for a new project).

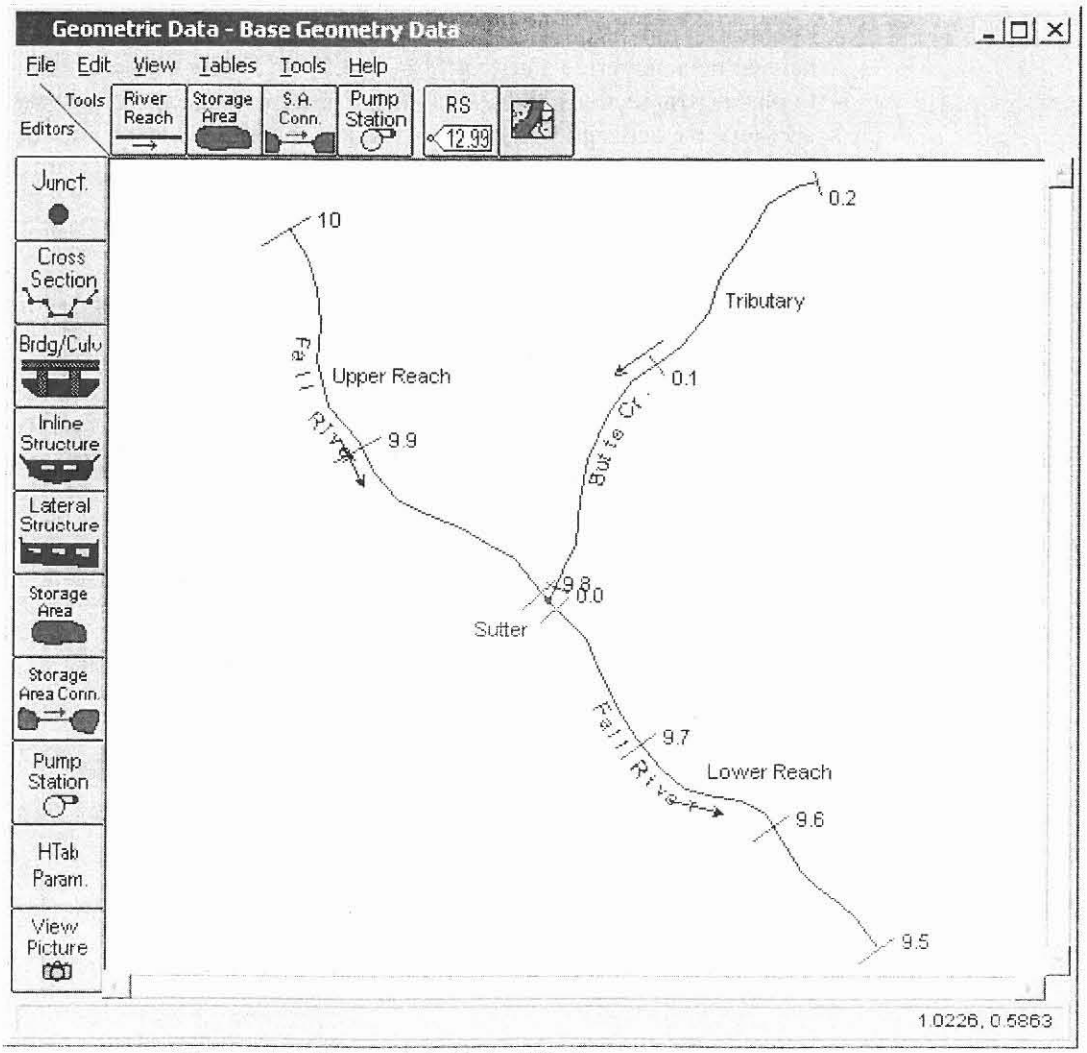


Figure 3.6 Geometric Data Window

The modeler develops the geometric data by first drawing in the river system schematic. This is accomplished, on a reach-by-reach basis, by pressing the **River Reach** button and then drawing in a reach from upstream to downstream (in the positive flow direction). After the reach is drawn, the user is prompted to enter a "River" and a "Reach" identifier. The River and reach identifiers can be up to 16 characters in length. As reaches are connected together, junctions are automatically formed by the interface. The modeler is also prompted to enter an identifier for each junction. For more information on developing the river system schematic, see Chapter 6 "Entering and Editing Geometric Data."

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 3.7. As shown, each cross section has a **River** name, **Reach** name, **River Station**, and a **Description**. The **River**, **Reach** and **River Station** identifiers are used to describe where the cross section is located in the river system. 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**.

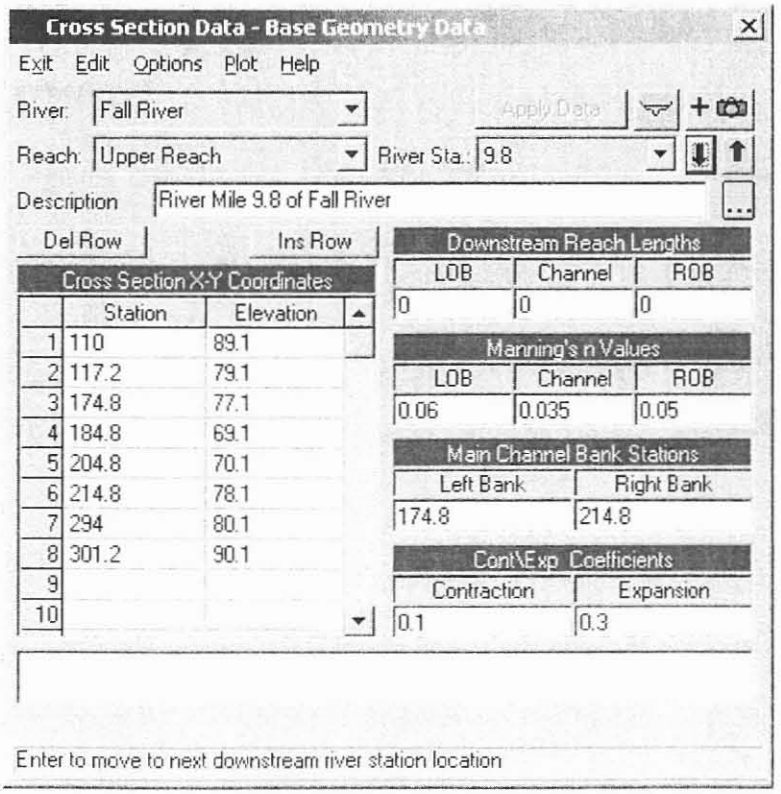


Figure 3.7 Cross Section Data editor

The basic data required for each cross section are shown on the Cross Section Data editor in Figure 3.7. 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; skew cross section; ineffective flow areas; levees; blocked obstructions; adding a lid to a cross section; add ice cover; add a rating curve; horizontal variation of n or k-values; and 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.

Entering Flow Data and Boundary Conditions

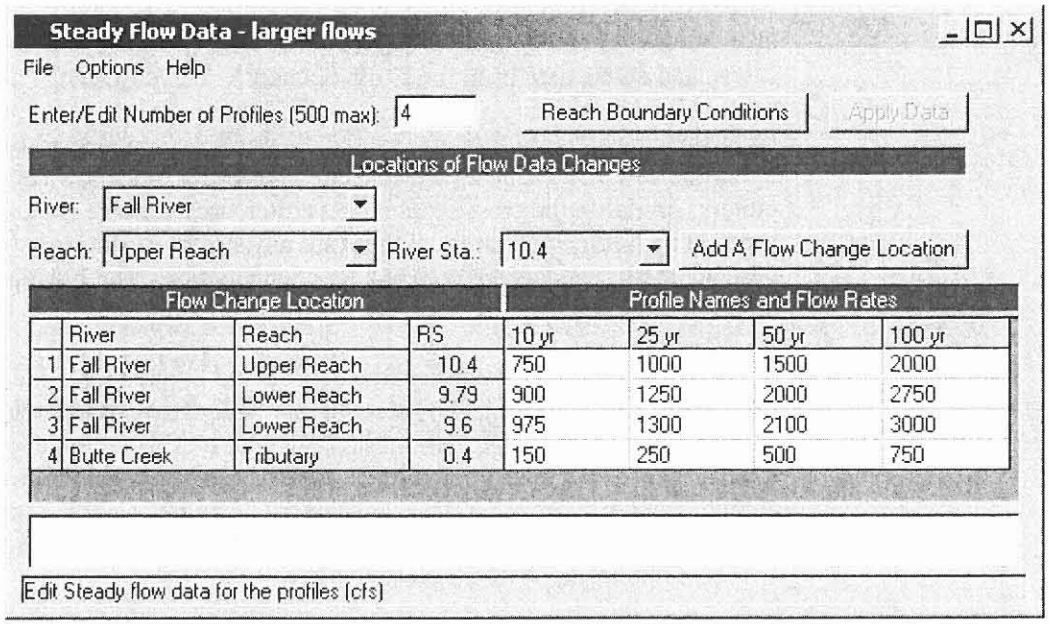
Once the geometric data are entered, the modeler can then enter either steady flow or unsteady flow data. The type of flow data entered depends upon the type of analyses to be performed. For the discussion in this chapter, it is assumed that a steady flow analysis will be performed. The data entry form for steady flow data is available under the **Edit** menu bar option on the HEC-RAS main window.

An example of the steady flow data entry form is shown in Figure 3.8, which is the **Steady Flow Data Editor**. As shown in Figure 3.8, 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. 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 from 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. This can be accomplished by selecting **Save Flow Data As** from the **File** option on the Steady Flow Data menu bar. Flow data is saved in a separate file. The user is only required to enter a title for the flow data, the filename is automatically assigned.



	River	Reach	RS	10 yr	25 yr	50 yr	100 yr
1	Fall River	Upper Reach	10.4	750	1000	1500	2000
2	Fall River	Lower Reach	9.79	900	1250	2000	2750
3	Fall River	Lower Reach	9.6	975	1300	2100	3000
4	Butte Creek	Tributary	0.4	150	250	500	750

Figure 3.8 Steady Flow Data window

Performing the Hydraulic Computations

Once all of the geometric data and flow data are entered, the modeler can begin to perform the hydraulic calculations. As stated previously, there are three types of calculations that can be performed in the current version of HEC-RAS: Steady Flow Analysis, Unsteady Flow Analysis, and Hydraulic Design Functions. The modeler can select any of the available hydraulic analyses from the **Run** menu bar option on the HEC-RAS main window. An example of the Simulation Manager window is shown in Figure 3.9, which is the Steady Flow Analysis window.

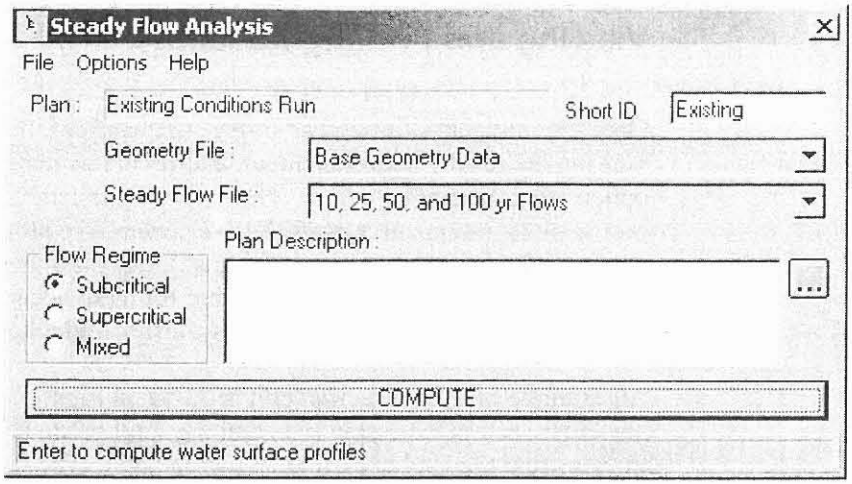


Figure 3.9 Steady Flow Analysis window

As shown in Figure 3.9, 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 for: performing a Floodway Encroachment Analysis; Setting locations for calculating flow distribution output; conveyance calculation options; friction slope methods; calculation tolerances; critical depth output; critical depth computation method; split flow optimization; data checking; setting log file levels; and viewing the log file output.

Once the modeler has selected a Plan and set all of the calculation options, the steady flow calculations can be performed by pressing the **Compute** button at the bottom of the Steady Flow Analysis window. When this button is pressed, the HEC-RAS system packages up all the data for the selected plan and writes it to a run file. The system then runs the steady flow model (SNET) and passes it the name of the run file. This process is executed in a separate window. Therefore, the modeler can work on other tasks while it is executing.

Viewing and Printing Results

Once the model has finished all of the computations, the modeler can begin viewing the results. Several output features are available under the **View** option from the main window. These options include: cross section plots; profile plots; rating curve plots; X-Y-Z perspective plots; hydrograph plot (if unsteady flow simulation was performed); tabular output at specific locations (Detailed Output Tables); tabular output for many locations (Profile Summary Tables); and the summary of errors, warnings, and notes.

An example of a cross section plot is shown in Figure 3.10. 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. These options include: zoom in; zoom out; full plot; pan; animate; selecting which plans, profiles and variables to plot; velocity distribution; viewing interpolated cross-sections; and control over the lines, symbols, labels, scaling, and grid options.

Hard copy outputs of the graphics can be accomplished in two different ways. Graphical plots can be sent directly from HEC-RAS to whichever printer or plotter the user has defined under the Windows Print Manager. Graphical 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.

An example of a profile plot is shown in Figure 3.11. 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.

An example of an X-Y-Z Perspective Plot is shown in Figure 3.12. 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.

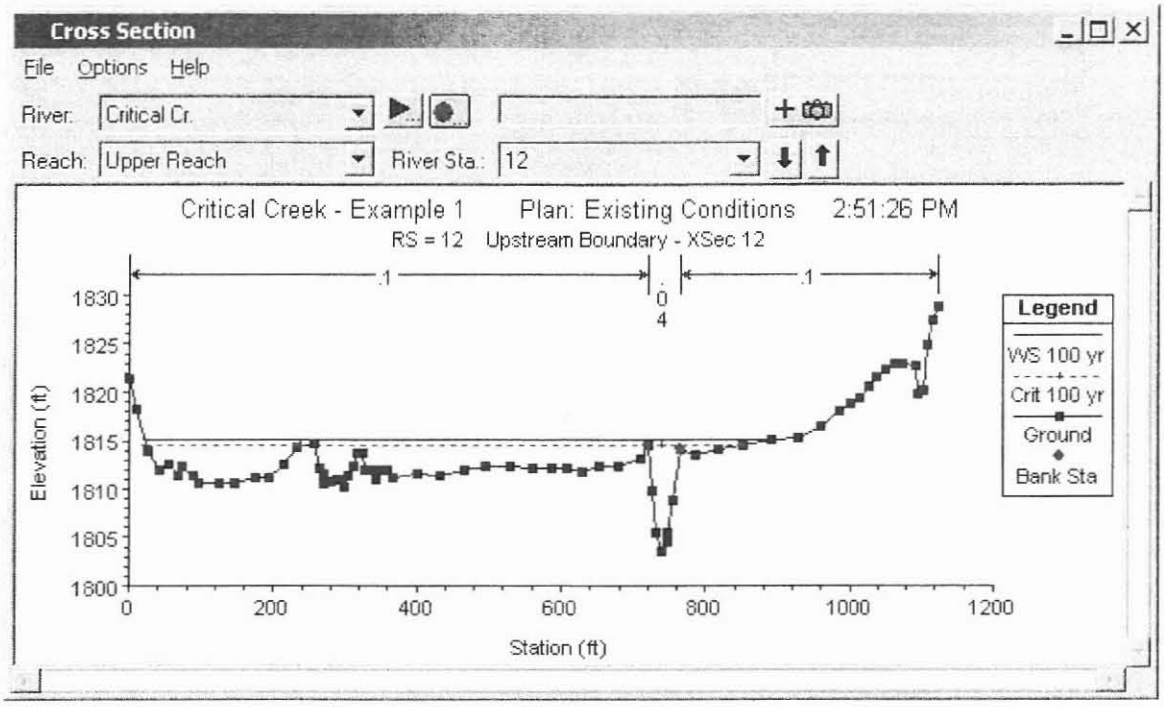


Figure 3.10 Cross Section Plot

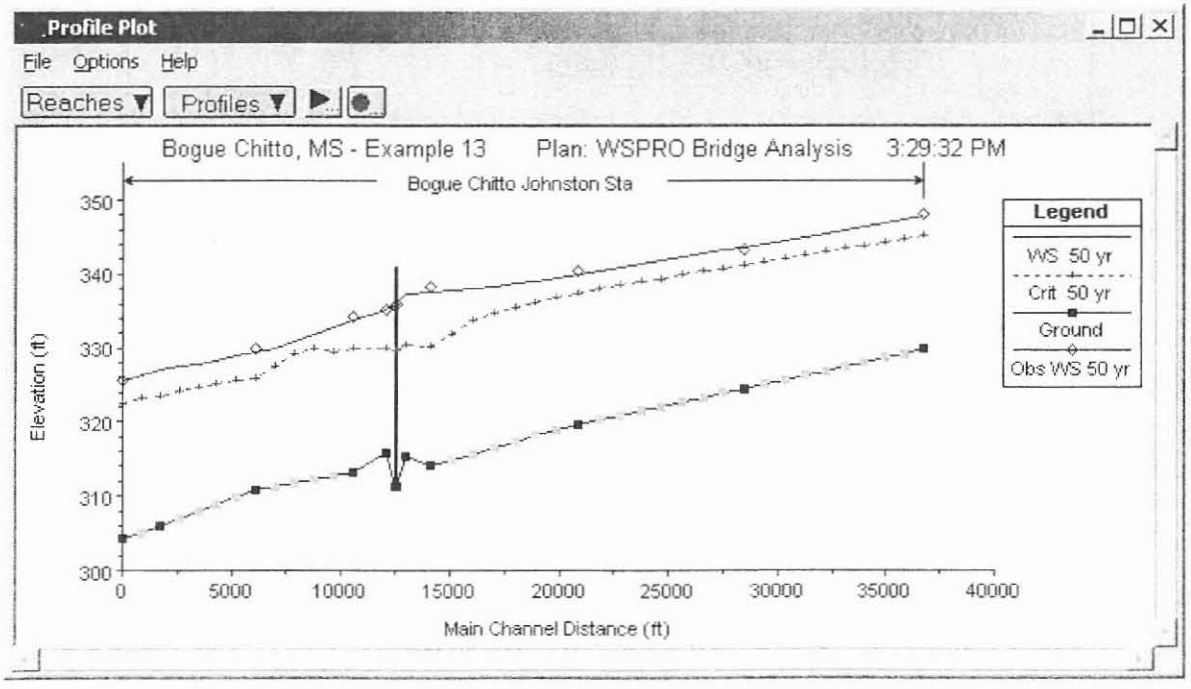


Figure 3.11 Profile Plot

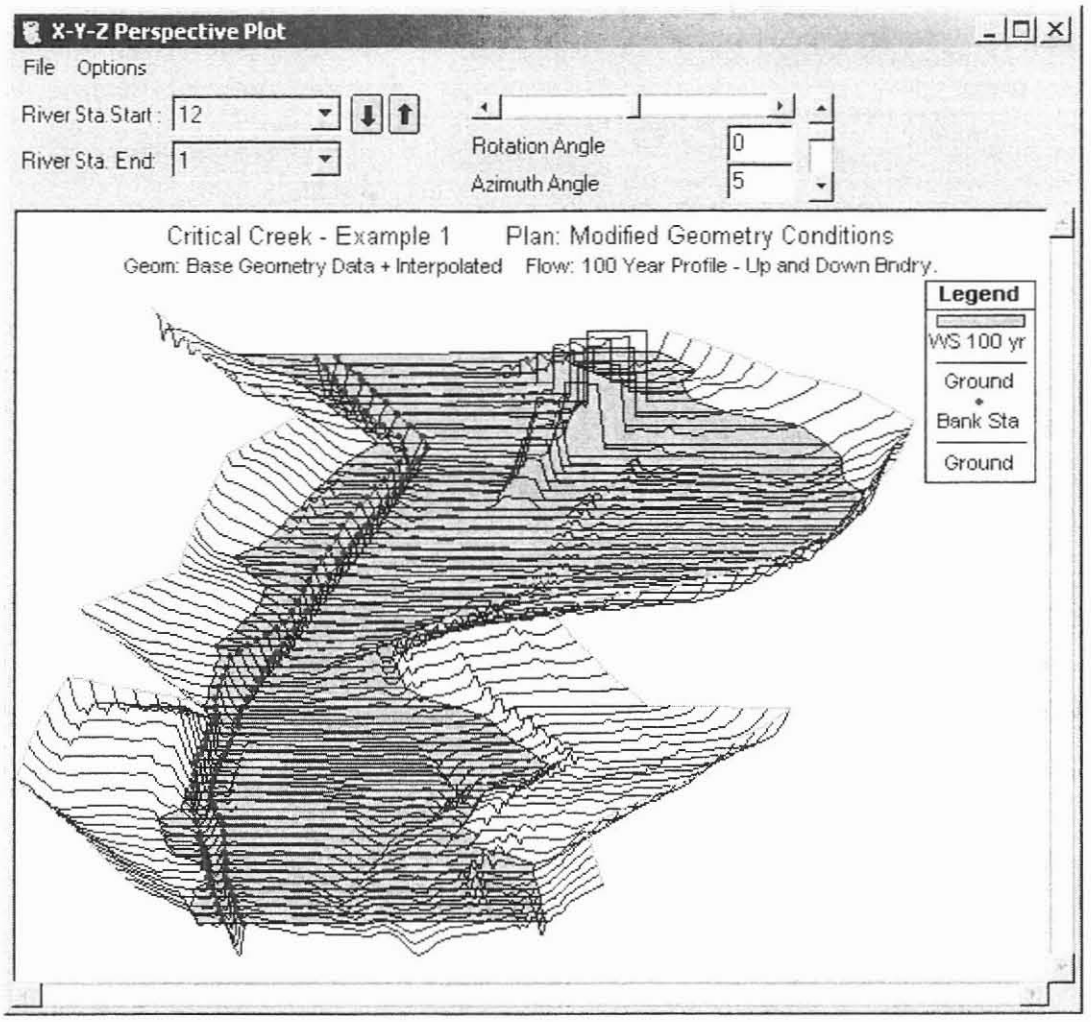
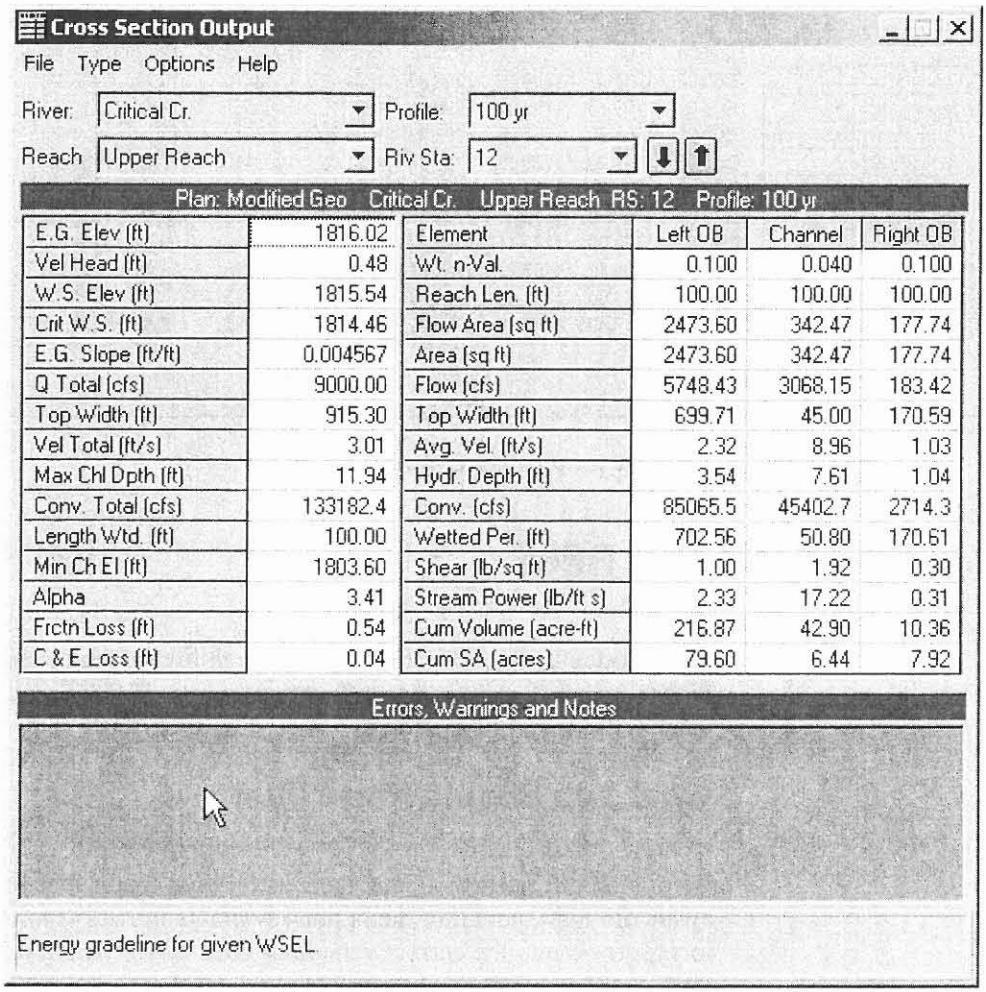


Figure 3.12 X-Y-Z Perspective Plot of River Reach with a Bridge

Tabular output is available in two different formats. The first type of tabular output provides detailed hydraulic results at a specific cross section location (Detailed Output Table). An example of this type of tabular output is shown in Figure 3.13.



		Element	Left OB	Channel	Right OB
E.G. Elev (ft)	1816.02				
Vel Head (ft)	0.48	Wt. n-Val.	0.100	0.040	0.100
W.S. Elev (ft)	1815.54	Reach Len. (ft)	100.00	100.00	100.00
Crit W.S. (ft)	1814.46	Flow Area (sq ft)	2473.60	342.47	177.74
E.G. Slope (ft/ft)	0.004567	Area (sq ft)	2473.60	342.47	177.74
Q Total (cfs)	9000.00	Flow (cfs)	5748.43	3068.15	183.42
Top Width (ft)	915.30	Top Width (ft)	699.71	45.00	170.59
Vel Total (ft/s)	3.01	Avg. Vel. (ft/s)	2.32	8.96	1.03
Max Chl Dpth (ft)	11.94	Hydr. Depth (ft)	3.54	7.61	1.04
Conv. Total (cfs)	133182.4	Conv. (cfs)	85065.5	45402.7	2714.3
Length Wtd. (ft)	100.00	Wetted Per. (ft)	702.56	50.80	170.61
Min Ch El (ft)	1803.60	Shear (lb/sq ft)	1.00	1.92	0.30
Alpha	3.41	Stream Power (lb/ft s)	2.33	17.22	0.31
Frctn Loss (ft)	0.54	Cum Volume (acre-ft)	216.87	42.90	10.36
C & E Loss (ft)	0.04	Cum SA (acres)	79.60	6.44	7.92

Figure 3.13 Tabular Detailed Output

The second type of tabular output shows a limited number of hydraulic variables for several cross sections and multiple profiles (Profile Summary Tables). An example of this type of tabular output is shown in Figure 3.14. There are several standard tables that are predefined 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.

Profile Output Table - Standard Table 1

File Options Std. Tables User Tables Locations Help

HEC-RAS Plan: Exist Cond River: Critical Cr. Reach: Upper Reach Profile: 100 yr Reload Data

Reach	River Sta	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)	
Upper Reach	12	9000.00	1803.60	1815.06	1814.46	1815.76	0.006851	10.51	2558.45	878.61	0.69
Upper Reach	11	9000.00	1800.70	1810.42	1810.42	1811.87	0.008552	12.03	1734.74	562.38	0.82
Upper Reach	10	9000.00	1794.40	1804.46	1803.69	1804.98	0.010276	10.48	2478.11	914.50	0.79
Upper Reach	9	9000.00	1788.70	1799.31	1799.31	1800.16	0.008851	11.48	2719.81	1216.82	0.80
Upper Reach	8	9500.00	1784.30	1793.89	1793.89	1795.08	0.008613	12.38	2524.66	1110.69	0.81
Upper Reach	7	9500.00	1777.20	1789.88	1788.87	1791.00	0.007410	13.16	2155.56	526.61	0.76
Upper Reach	6	9500.00	1774.50	1784.29	1784.29	1786.35	0.011143	13.38	1266.30	332.38	0.93
Upper Reach	5	9500.00	1768.50	1776.81	1776.81	1778.18	0.013216	13.55	1830.26	583.34	0.97
Upper Reach	4	9500.00	1763.00	1773.44	1772.23	1773.88	0.004991	9.32	2988.72	760.42	0.59
Upper Reach	3	9500.00	1759.40	1767.29	1765.75	1769.34	0.019810	16.09	1610.99	621.76	1.20
Upper Reach	2	9500.00	1753.60	1761.54	1760.03	1762.10	0.009413	10.36	2323.62	682.71	0.79
Upper Reach	1	9500.00	1747.40	1756.71	1755.71	1757.21	0.010002	9.91	2403.99	728.01	0.79

Figure 3.14 Profile Output Table

Importing HEC-2 Data

An important feature of HEC-RAS is the ability to import HEC-2 data. This feature makes it easy for a user to import existing HEC-2 data sets and start using HEC-RAS immediately.

What You Should Know First

Before importing HEC-2 data, there are several things that you should be aware of. First, not all of the options available in HEC-2 have been incorporated into the current version of HEC-RAS. The following is a list of HEC-2 options that are not available in the current version of HEC-RAS:

- Compute Manning's n from high water marks (J1)
- Archive (AC)
- Free Format (FR)
- Storage Outflow for HEC-1 (J4)

HEC-2 data sets containing these options can still be imported, but these data options will be ignored.

Another important issue to be aware of is how the cross sections are identified. In HEC-RAS, each cross section is identified with a River name, Reach name, and a River Station. The river stationing must be in order from highest river stationing upstream to lowest river stationing downstream. When the user goes to import HEC-2 data, a pop up window will appear (Figure 3.15), asking the user to select a method for identifying the river stationing of the cross sections. If you select "Use HEC-2 Section ID's," the program will use the first field of the X1 record for the river stationing of the cross section. If you choose this method, you must be sure that the cross sections in the HEC-2 file are numbered with highest river stationing upstream, and that no two cross sections have the same river station identifier. If these two requirements are not met, the program will not import the data correctly. An alternative is to select "Use Sequential Counter." This method simply assigns river stations as 1, 2, 3, etc. in the order in which the cross sections are found in the HEC-2 file (still maintaining highest numbers upstream and lowest numbers downstream.

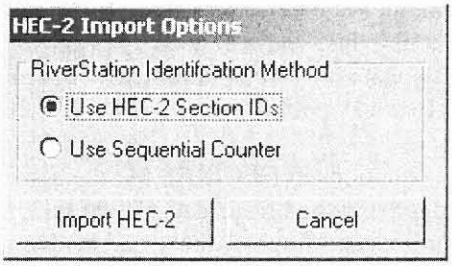


Figure 3.15 Method for Identifying River Stations from HEC-2 Data

After the HEC-2 data is imported into HEC-RAS, you may need to make some modifications to the data. HEC-RAS is a completely new program. As HEC-RAS was being developed, we tried to improve the hydraulic computations in every way we could. Some of these improvements have made it necessary to get more information and/or different information from the user for a specific type of computation. The following is a list of features in which the data requirements for HEC-2 and HEC-RAS have changed, and it **may be** necessary for the user to modify the data after it is imported:

- Special Bridge (SB)
- Special Culvert (SC)
- Normal Bridge (X2, BT)
- Encroachments and Floodway Determination (X3, ET)
- Ineffective Flow Areas (X3)

When bridge data are imported, the user must take special care to ensure that the data are correctly representing the bridge. The bridge routines in HEC-RAS are more detailed than HEC-2, and therefore you may have to modify some data and/or enter some additional data. Whenever you import an HEC-2 data set with bridge data, carefully review all the data for each bridge. Chapter 6 of this user's manual describes the required data for bridges in HEC-RAS. **Appendix C of the HEC-RAS Hydraulic Reference Manual contains a detailed discussion of the computational differences between HEC-RAS and HEC-2**. Some key differences between the bridge routines of HEC-2 and HEC-RAS are as follows:

1. **Special Bridge Data Sets**

HEC-RAS does not use a trapezoidal approximation for low flow through the bridge opening. The actual bridge opening is used in both the Yarnell method and the momentum method. This could be a problem for HEC-2 special bridge data sets that do not include low chord information on the BT data. If you have a data set like this, you will need to modify the bridge deck information after the data have been imported. This can be done from the HEC-RAS Deck/Roadway editor.

The pressure flow equations in HEC-RAS use the actual bridge opening, defined by the ground and the bridge data. In HEC-2, the user was required to enter an area for pressure flow. If the actual bridge opening produces a different area than what the user had entered in the HEC-2 data deck, the program will get different results for pressure flow, and pressure and weir flow answers.

Pier information from the SB record is incorporated as a single pier in the HEC-RAS data set (this is how it was treated in HEC-2). Piers are treated as separate pieces of data in HEC-RAS. For special bridges that have piers, you may want to change the single pier to multiple piers, depending on what is actually at the bridge. Pier information can be modified using the **Pier** editor.

2. **Normal Bridge Data Sets**

Because piers are treated as a separate piece of data in HEC-RAS, they must not be included in the cross section data or the bridge deck. Since it is common to include pier information as part of the cross section or bridge deck in HEC-2, these data will need to be modified. For data sets that have piers, you will need to remove the pier information from the cross section or bridge deck, and then add the information back in using the **Pier** editor.

Steps for Importing HEC-2 Data

To import HEC-2 data, do the following:

1. Start a new project by selecting **New Project** under the **File** menu option on the HEC-RAS main window (Figure 3.16). When this option is selected a window will appear allowing you to select the drive and directory for the new project, then enter a project title and filename. Press the OK button, and then a pop up window will appear asking you to confirm the information.

2. Select the **Import HEC-2 Data** option under the **File** menu on the main window (Figure 3.16). A pop up window will appear (Figure 3.17), which will allow you to select a drive, path, and filename for the HEC-2 data file. In addition to the filenames being listed, the first line of each HEC-2 data file is shown under the title field on the window. Once you have selected the file you want, press the **OK** button.

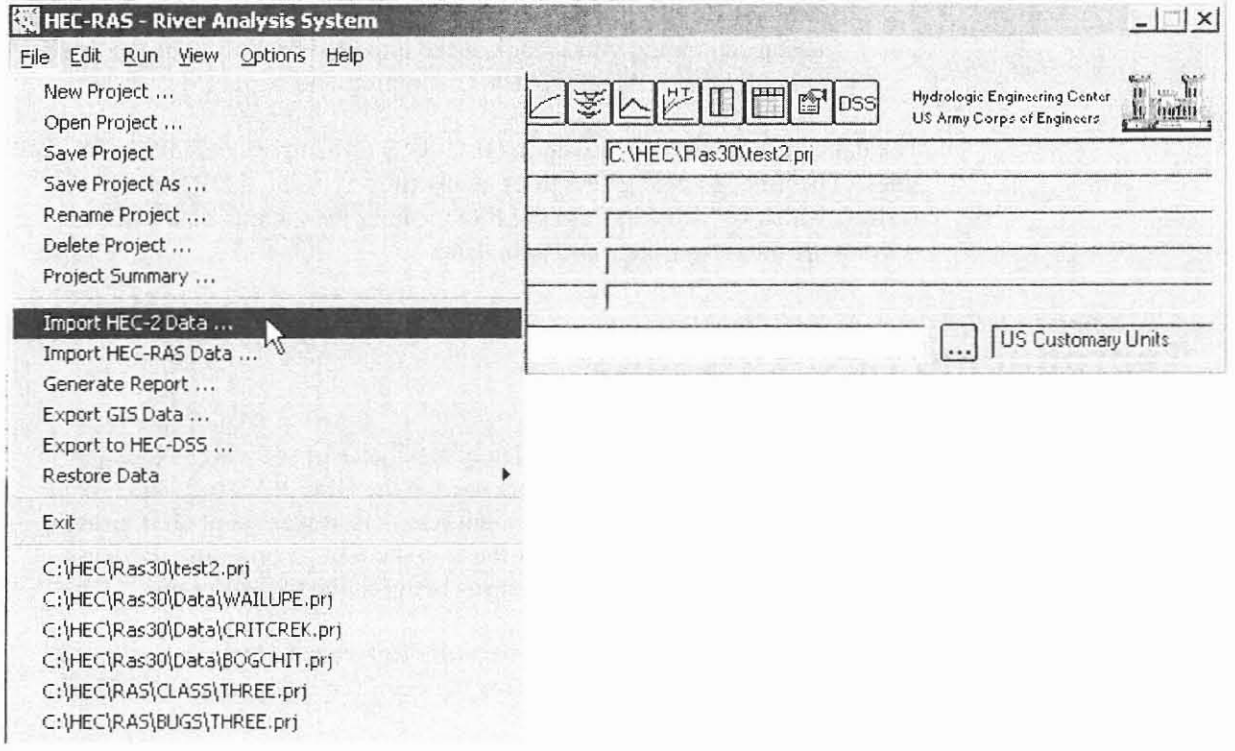


Figure 3.16 HEC-RAS Main Window With File Menu Options Shown

3. Once you have selected an HEC-2 file and pressed the **OK** button, a pop up window will appear asking you to select a method for identifying the river stationing of the cross sections (this was discussed under the "What You Should Know First" section). Select a method and press the **Import HEC-2** button.

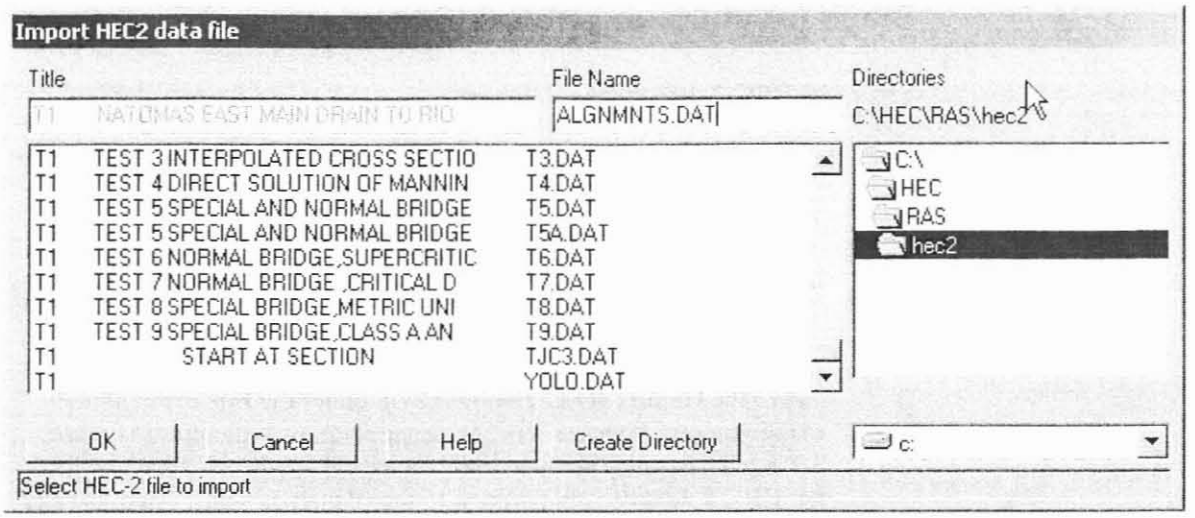


Figure 3.17 Window for Importing HEC-2 Data

4. If the HEC-2 data file contains any bridges or culverts, a note will appear reminding you to look at the imported data of all of the bridges and culverts to ensure the data is complete and correct.

The data are automatically saved in HEC-RAS format with default names and titles. The user can change the titles at any time by using the **Rename** feature, which is available from the **File** menu of the various data editors (Geometric data, flow data, and plan data).

Reproducing HEC-2 Results

The HEC-RAS program is a completely new piece of software. None of the hydraulic routines from HEC-2 were used in the HEC-RAS software. When HEC-RAS was being developed, a significant effort was spent on improving the computational capabilities over those in the HEC-2 program. Because of this, there are computational differences between the two programs.

Appendix C, of the HEC-RAS Hydraulic Reference Manual, outlines in detail the computational differences between the two programs. Please review this closely!

When importing HEC-2 data, and attempting to reproduce the results of a previous study, the following is a list of items that should be considered:

1. First, is the data that you have imported good data? In other words, did it come from a working HEC-2 model, and was that model considered as being hydraulically sound. Are there an adequate number of cross sections? And are there any mistakes in the cross section data? Review the data closely, before you assume that it is good!

2. The default method for calculating conveyance in HEC-RAS is different from HEC-2. However, HEC-RAS has the ability to compute conveyance with the HEC-2 methodology. If you are trying to reproduce HEC-2 results, you may want to switch HEC-RAS to the HEC-2 method of computing conveyance. To do this, from the Steady Flow Analysis window select **Options** from the menu bar then select **Conveyance Calculations**. When this is selected, a pop up window will appear as shown in Figure 3.18. There are two options available, the HEC-RAS default method (break in n-value method) and the HEC-2 style method. Select the HEC-2 style method if you are trying to reproduce HEC-2 results. For more information on the differences in conveyance calculations, see Appendix C of the HEC-RAS Hydraulic Reference manual.

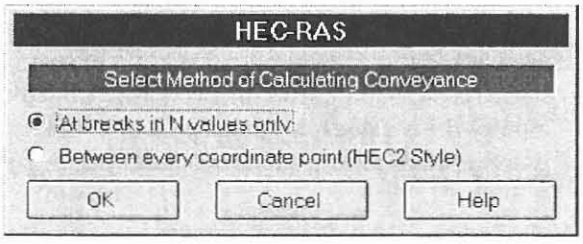


Figure 3.18 HEC-RAS Conveyance Calculation Methods

3. The HEC-RAS bridge routines are more comprehensive than the HEC-2 bridge routines, and therefore differences can occur at bridge locations. First, review the bridge data closely and make sure it accurately represents the bridge you are trying to model. If you feel it is necessary to match the results of a previous study at the bridge, then your only alternative is to adjust the coefficients that are being used in the bridge modeling approach (i.e., pressure and weir flow coefficients, low flow coefficients, contraction and expansion coefficients, etc...). For detailed information on the differences between the HEC-RAS and HEC-2 bridge routines, please review Appendix C of the Hydraulic Reference manual.

4. Sometimes differences can occur at locations where the programs have defaulted to a critical depth solution. First you should ask yourself if critical depth is an appropriate solution for this location. It is a common problem for both programs to default to critical depth when the cross sections are spaced too far apart. If you feel critical depth is an appropriate solution, then in general the HEC-RAS answer will be better than HEC-2. The critical depth routines in HEC-RAS are much more comprehensive than HEC-2. HEC-RAS has tighter error limits for locating critical depth, as well as the ability to find multiple critical depths and detect which is the most appropriate.

5. Differences can also occur at locations where floodway encroachments are being computed. The HEC-RAS floodway encroachment routines have been improved over those available in HEC-2. Also, the default at bridges in HEC-RAS is to perform the encroachment analysis, while the default in HEC-2 was to not encroach at bridges. For more details on differences between encroachment routines, please review Appendix C of the HEC-RAS Hydraulic Reference Manual.

6. After carefully reviewing items one through five above, if you still have computational differences in the computed profiles, you may need to modify Manning's n values in order to reproduce the previous study results. In general, this is not suggested. If you do decide to modify the n values, try to keep them within a realistic range of what is appropriate for the stream you are working on.

Getting and Using Help

On-line help is available from within the HEC-RAS software. Help can be accessed by selecting the **Help** menu option at the top of each window, or by pressing the **F1** function key.

CHAPTER 4

Example Application

This chapter provides an example application of how to perform steady flow water surface profile calculations with HEC-RAS. The user is taken through a step-by-step procedure of how to enter data, perform calculations, and view the results.

In order to get the most out of this chapter, you should perform each of the steps on your own computer. Also, before you try the example application, you should have read the first three chapters in this manual.

Contents

- Starting a New Project
- Entering Geometric Data
- Entering Steady Flow Data
- Performing the Hydraulic Calculations
- Viewing Results
- Printing Graphics and Tables
- Exiting the Program

Starting a New Project

To begin this example, let's first start the HEC-RAS program. Double click the HEC-RAS icon in Windows. The main window should appear as shown in Figure 4.1 (except yours will be blank the first time you start the program).

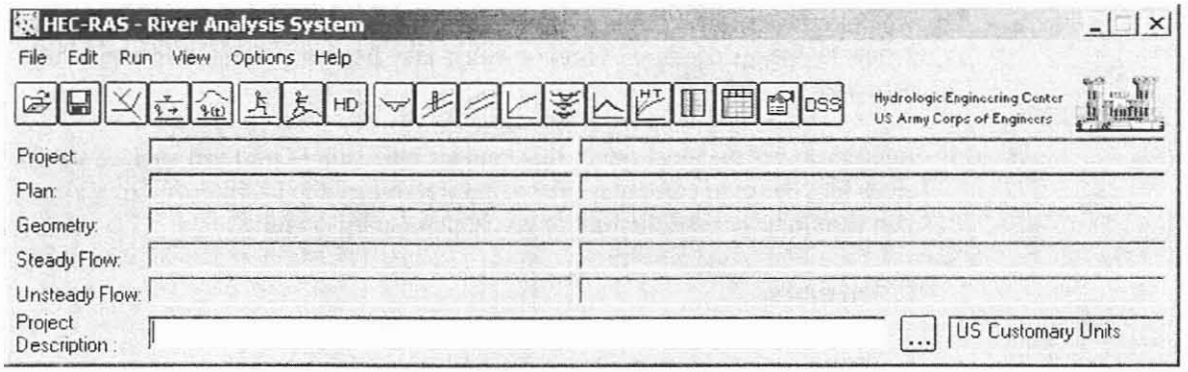


Figure 4.1 HEC-RAS Main Window

The first step in developing an HEC-RAS application is to start a new project. Go to the **File** menu on the main window and select **New Project**. The New Project window should appear as shown in Figure 4.2 (except the title and file name fields will be blank when it first comes up).

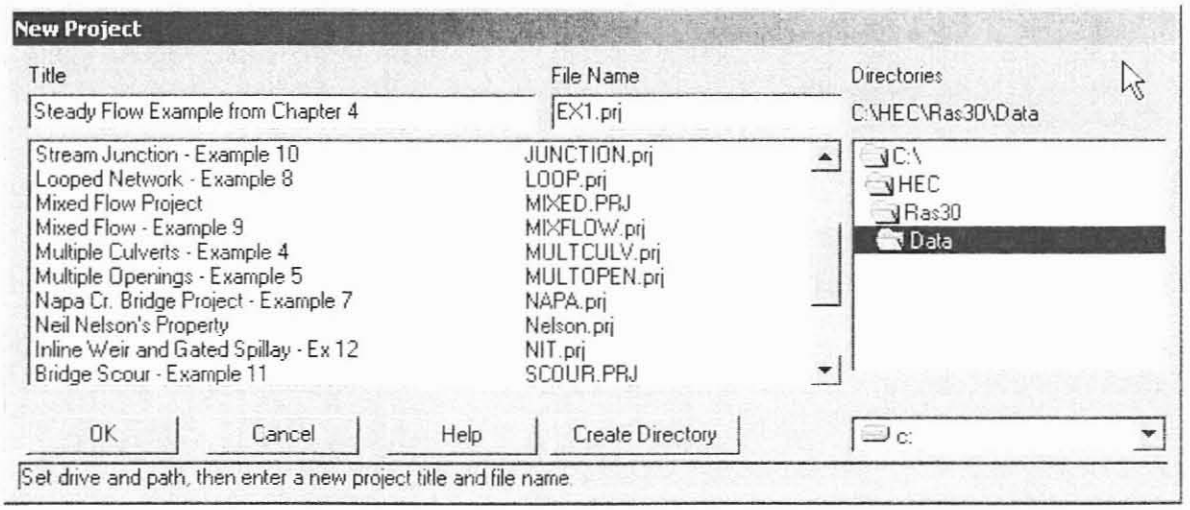


Figure 4.2 New Project Window

First set the drive (e.g., C:) and the directory that you would like to work in. Next enter the project title and filename as shown in Figure 4.2. Once you have entered the information, press the **OK** button to have the data accepted.

Entering Geometric Data

The next step in developing a steady flow model with HEC-RAS is to enter the geometric data. This is accomplished by selecting **Geometric Data** from the **Edit** menu on the HEC-RAS main window. Once this option is selected the geometric data window will appear, except yours will be blank when you first bring it up (Figure 4.3).

Drawing the Schematic of the River System

In this example we are going to develop a two-river (three hydraulic reaches) system as shown in Figure 4.3. Draw the river system schematic by performing the following steps:

1. Click the **River Reach** button on the geometric data window.

2. Move the mouse pointer over to the drawing area and place the pointer at the location in which you would like to start drawing the first reach.

3. Press the left mouse button once to start drawing the reach. Move the mouse pointer and continue to press the left mouse button to add additional points to the line segment. To end the drawing of the reach, double click the left mouse button and the last point of the reach will be placed at the current mouse pointer location. All reaches must be drawn from upstream to downstream (in the positive flow direction), because the program assumes this to be true.

4. Once the reach is drawn, the interface will prompt you to enter an identifier for the **River** name and the **Reach** name. The River identifier can be up to 32 characters, while the reach name is limited to 12 characters. In this example, there is one river named **Fall River** and another one named **Butte Cr**. Fall river contains two hydraulic reaches, which are labeled **Upper Reach** and **Lower Reach**. Butte Cr. has been entered as a single hydraulic reach, and the reach name is **Tributary**.

5. Repeat steps one through four for each reach. After you enter the identifiers for Butte Cr., you will also be prompted to enter an identifier for the junction. Junctions in HEC-RAS are locations where two or more reaches join together or split apart.

Once you have finished drawing in the river system, there are several options available for editing the schematic. These options include: change name, move object (objects are labels, junctions, and points in the reaches), add points to a reach, remove points from a reach, delete a reach, and delete a junction. The editing features are located under the **Edit** menu on the Geometric Data window. **Note: when you first draw your schematic there will not be any tic marks representing cross sections as shown in Figure 4.3. The tic marks only show up after you have entered cross sections.**

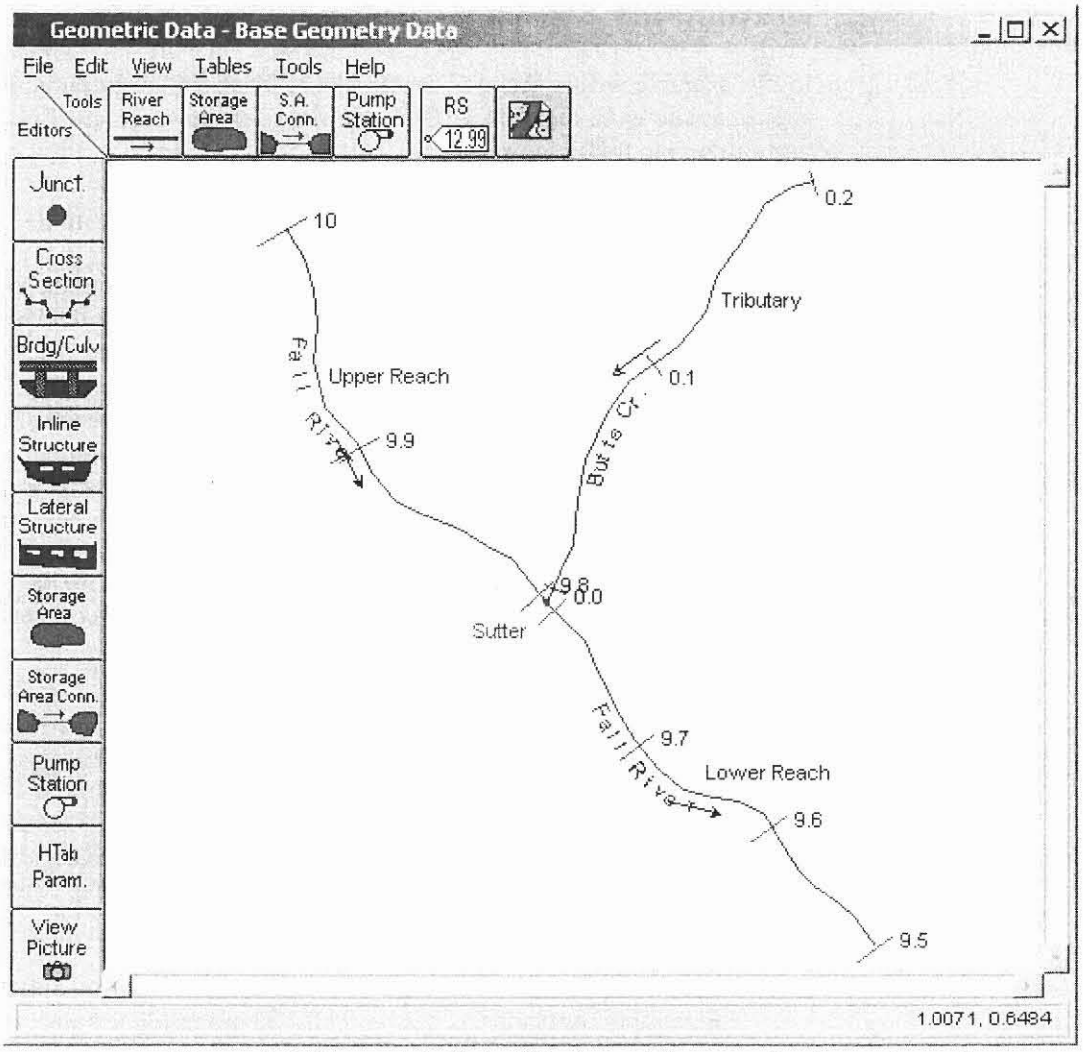


Figure 4.3 Geometric Data Window with example river schematic

Entering Cross Section Data

The next step is to enter the cross section data. This is accomplished by pressing the **Cross Section** button on the Geometric Data window (Figure 4.3). Once this button is pressed, the Cross Section Data editor will appear as shown in Figure 4.4 (except yours should be blank). To enter cross section data do the following:

1. Select a **River** and a **Reach** to work with. For this example start with the Fall River, Upper Reach.

2. Go to the **Options** menu and select **Add a new Cross Section**. An input box will appear to prompt you to enter a river station identifier

   for the new cross section. The identifier does not have to be the actual river station, but it must be a numeric value. The numeric value describes where this cross section is located in reference to all the other cross sections within the reach. Cross sections are located from upstream (highest river station) to downstream (lowest river station). For this cross section enter a value of 10.0.

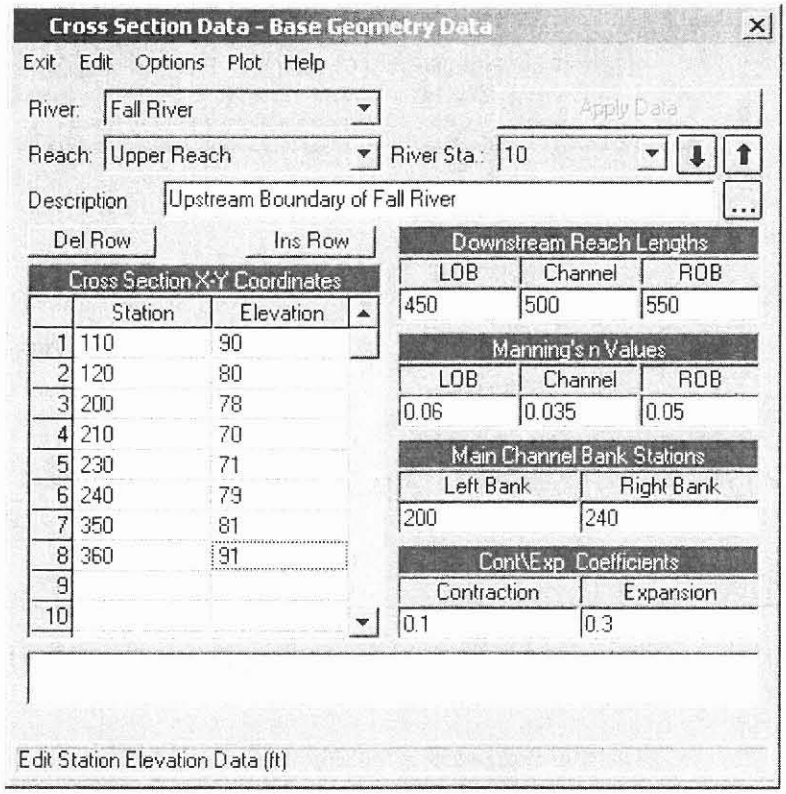


Figure 4.4 Cross Section Data Editor with example data

3. Enter all of the data for this cross section as it is shown in Figure 4.4.

4. Once all the data are entered press the **Apply Data** button. This button is used to tell the interface that you want the data to be accepted into memory. This button does not save the data to your hard disk, which can only be accomplished from the **File** menu on the Geometric Data window.

5. Plot the cross section to visually inspect the data. This is accomplished by pressing the **Plot Cross Section** option under the

Plot menu on the Cross Section Data Editor. The cross section should look the same as that shown in Figure 4.5.

In general, the five steps listed would be repeated for every cross section that is entered. In order to reduce the amount of data entry for this example, the current cross section will be copied and adjusted to represent other cross sections within the river system.

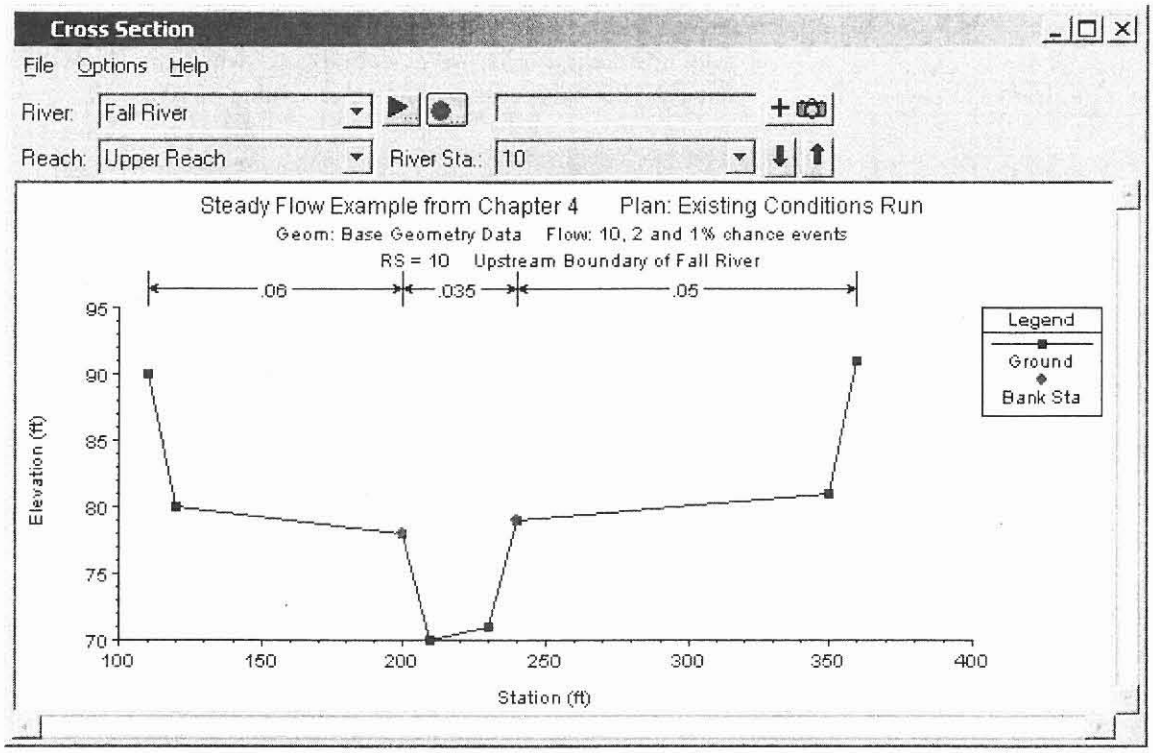


Figure 4.5 Cross Section Plot for river mile 10.0 of Fall Creek

The following steps should be followed to copy the current cross section:

1. Go to the **Options** menu on the Cross Section Data Editor and select **Copy Current Cross Section**. An input box will appear to prompt you to select a river and a reach, and then enter a river station for the new cross section. For this example, keep the river and reach as Fall River and Upper Reach, then enter a new river station of 9.9. Press the **OK** button and the new cross section will appear in the editor.

2. Change the cross section description to "River Mile 9.9 of Fall River."

3. Adjust all the elevations of the cross section by -0.5 feet. This is accomplished by selecting the **Adjust Elevations** feature from the **Options** menu on the Cross Section Data Editor.

4. Adjust the cross section stationing to reduce the overbanks by 10%.

This is accomplished by selecting the **Adjust Stations** feature from the **Options** menu on the Cross Section Data Editor, then select **Multiply by a Factor**. When the input box appears for this option, three data entry fields will be available to adjust the stationing of the left overbank, channel, and the right overbank separately. Enter values of 0.90 for the right and left overbanks, but leave the main channel field blank. This will reduce the stationing of both overbanks by 10%, but the main channel will not be changed.

5. Downstream reach lengths remain the same for this cross section.

6. Press the **Apply Data** button.

7. Plot the cross section to visually inspect it.

These seven steps should be repeated to enter all the data for Fall River (Upper and Lower Reach). The necessary adjustments are listed in Table 4.1. Perform the cross section duplications in the order that they are listed in the table. Make sure to change the description of each cross section, and also press the **Apply Data** button after making the adjustments for each cross section.

Table 4.1 Cross Section adjustments for duplicating sections

Cross Section		Adjusted Elevation	Adjusted Stationing			Downstream Reach Lengths		
Reach	River Sta.		Left O.B.	Channel	Right O.B.	Left O.B.	Channel	Right O.B.
Upper	9.8	-0.4	0.80	-	0.80	0.0	0.0	0.0
Lower	9.79	-0.1	1.20	1.20	1.20	500	500	500
Lower	9.7	-0.5	1.20	1.20	1.20	500	500	500
Lower	9.6	-0.3	-	-	-	500	500	500
Lower	9.5	-0.2	-	-	-	0.0	0.0	0.0

This completes all the cross section data for Fall River (upper and Lower reach). Now let's work on entering the data for the Butte Creek tributary. To enter the first cross section in the Butte Creek tributary do the following:

1. Go to the **River** text box on the Cross Section Data Editor and select the **Butte Cr.** river. The Reach of "Tributary" will automatically be selected since it is the only reach in Butte Creek.

2. Select **Add a new Cross Section** from the **Options** menu. When the popup box appears to prompt you to enter a new river station, enter a value of **0.2**.

3. Enter all the data for this cross section as shown in Figure 4.6.

4. Once all the data are entered for this section, press the **Apply Data** button.

5. Plot the cross section to inspect the data.

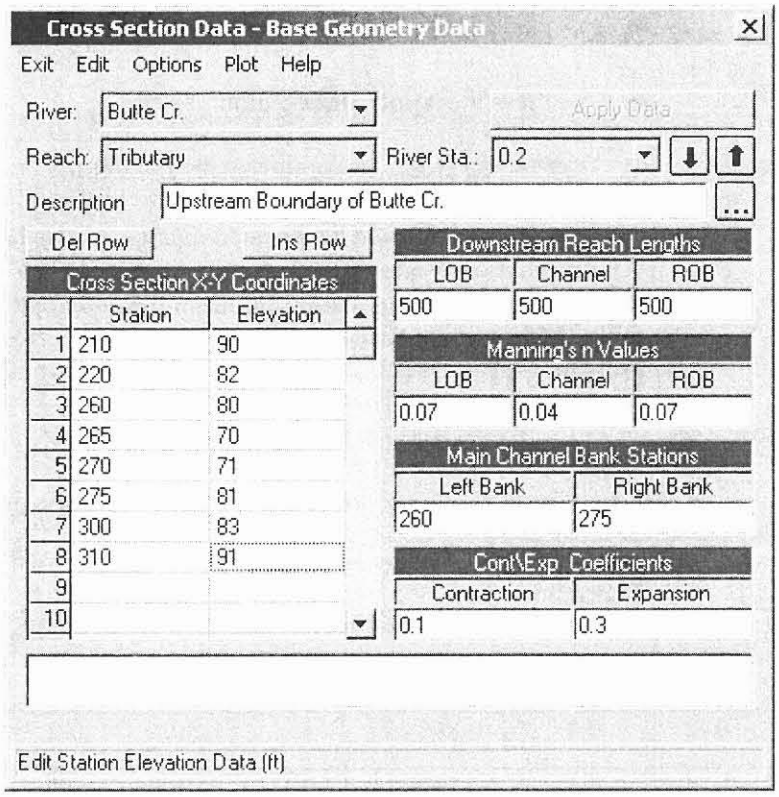


Figure 4.6 Cross Section Editor with river mile 0.2 of Butte Creek

There are two other cross sections that need to be developed for the Butte Creek tributary. These two cross sections will be developed by duplicating the cross section that you just entered, and then adjusting the elevations and stationing. The necessary adjustments are listed in Table 4.2. Perform the cross section adjustments in the order that they are listed in the table. Make sure to change the description of each cross section and press the **Apply Data** button after editing is complete.

Table 4.2 Cross Section adjustments for Butte Creek sections

Cross Section		Adjusted Elevation	Adjusted Stationing			Downstream Reach Lengths		
Reach	River Sta.		Left O.B.	Channel	Right O.B.	Left O.B.	Channel	Right O.B.
Butte Cr.	0.1	-0.6	-	-	-	500	500	500
Butte Cr.	0.0	-0.3	-	-	-	0.0	0.0	0.0

Now that all of the cross section data are entered, save the data to a file before continuing. Saving the data to a file is accomplished by selecting the "**Save Geometry Data As**" option from the **File** menu on the Geometric Data window. After selecting this option you will be prompted to enter a Title for the geometric data. Enter "Base Geometry Data" for this example, then press the **OK** button. A file name is automatically assigned to the geometry data based on what you entered for the project filename.

Entering Junction Data

The next step is to enter the junction data. Junction data consist of a description, and reach lengths across the junction. In this example there is only one junction, which is labeled **Sutter**. To enter Junction data, press the **Junction** button on the Geometric Data window. Enter the junction data as shown in Figure 4.7.

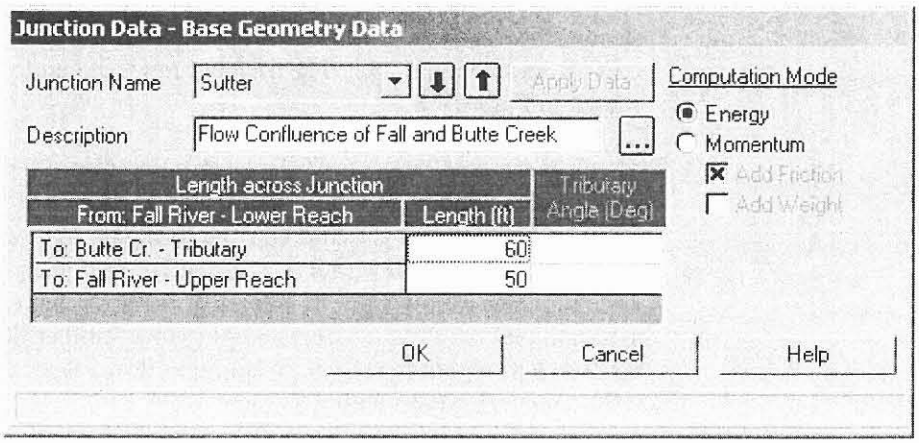


Figure 4.7 Junction Data Editor, with Sutter junction data

Reach lengths across the junction are entered in the junction editor, rather than in the cross section data. This allows for the lengths across very complicated confluences (i.e., flow splits) to be accommodated. In the cross section data, the reach lengths for the last cross section of each reach should be left blank or set to zero.

In this example the energy equation will be used to compute the water surface profile through the junction. If the momentum equation is selected, then an angle can be entered for one or more of the reaches flowing into or out of a junction. The momentum equation is set up to account for the angle of the flow entering the junction.

Once you have all of the data entered for the junction, apply the data and close the window by pressing the **OK** button.

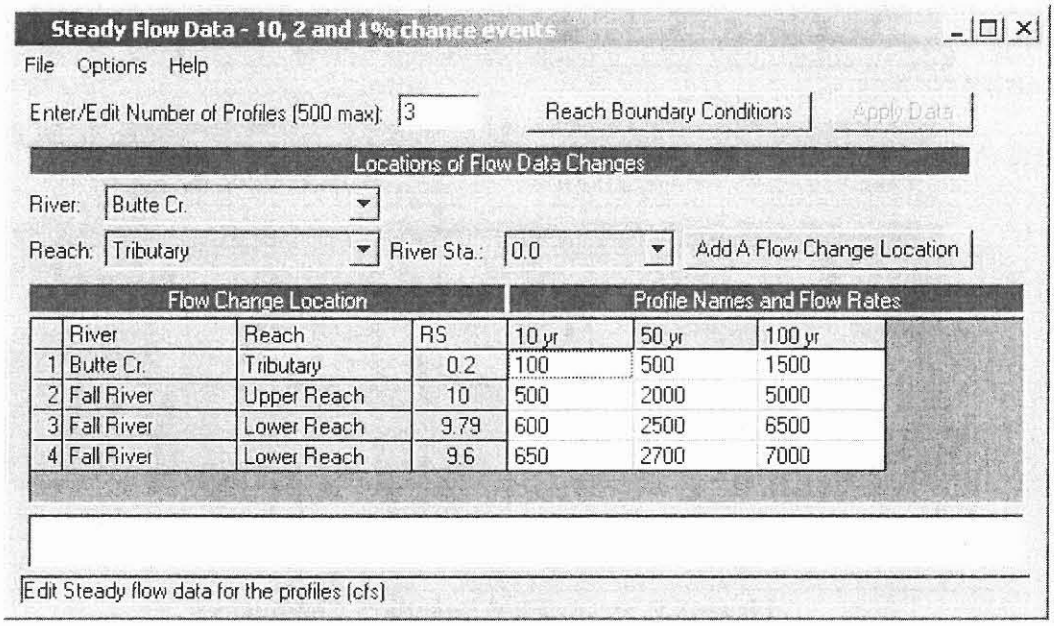
Saving the Geometry Data

At this point in the example, all of the geometric data has been entered. Before we continue with the example, you should save the geometric data to the hard disk. Since the data have already been saved once, you simply have to select **Save Geometry Data** from the **File** menu on the Geometric Data window. We can now go on to enter the Steady Flow data.

Entering Steady Flow Data

The next step in developing the required data to perform steady flow water surface profile calculations is to enter the steady flow data. To bring up the steady flow data editor, select **Steady Flow Data** from the **Edit** menu on the HEC-RAS main window. The Steady Flow Data editor should appear as shown in Figure 4.8.

The first piece of data to enter is the number of profiles to be calculated. For this example, enter "3" as shown in Figure 4.8. The next step is to enter the flow data. Flow data are entered from upstream to downstream for each reach. At least one flow rate must be entered for every 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 reach. Additional flow values can be entered at any cross section location within a reach.



	River	Reach	RS	10 yr	50 yr	100 yr
1	Butte Cr.	Tributary	0.2	100	500	1500
2	Fall River	Upper Reach	10	500	2000	5000
3	Fall River	Lower Reach	9.79	600	2500	6500
4	Fall River	Lower Reach	9.6	650	2700	7000

Figure 4.8 Steady Flow Data Editor, with example problem data

In this example, flow data will be entered at the upstream end of each reach. An additional flow change location will be entered at river mile 9.6 of the Fall River in the Lower Reach. To add an additional flow change location into the table, first select the Fall River, Lower Reach from the **Reach** list box. Next, select the desired river station location (9.6 in this example) from the **River Sta.** list box. Finally, press the **Add a Flow Change Location** button. The new flow location should appear in the table. Now enter all of the flow data into the table as shown in Figure 4.8. Profile labels will automatically default to "PF#1," "PF#2," etc. You can change these labels to whatever you want. In this example they have been changed to "10 yr," "50 yr," and "100 yr," to represent the statistical return period of each of the events being modeled.

The next step is to enter any required boundary conditions. To enter boundary conditions, press the **Enter Boundary Conditions** button at the top of the Steady Flow Data editor. The boundary conditions editor will appear as shown in Figure 4.9, except yours will be blank the first time you open it.

Boundary conditions are necessary to establish the starting water surface at the ends of the river system. A starting water surface is necessary in order for the program to begin the calculations. In a subcritical flow regime, boundary conditions are only required 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 open ends of the river system.

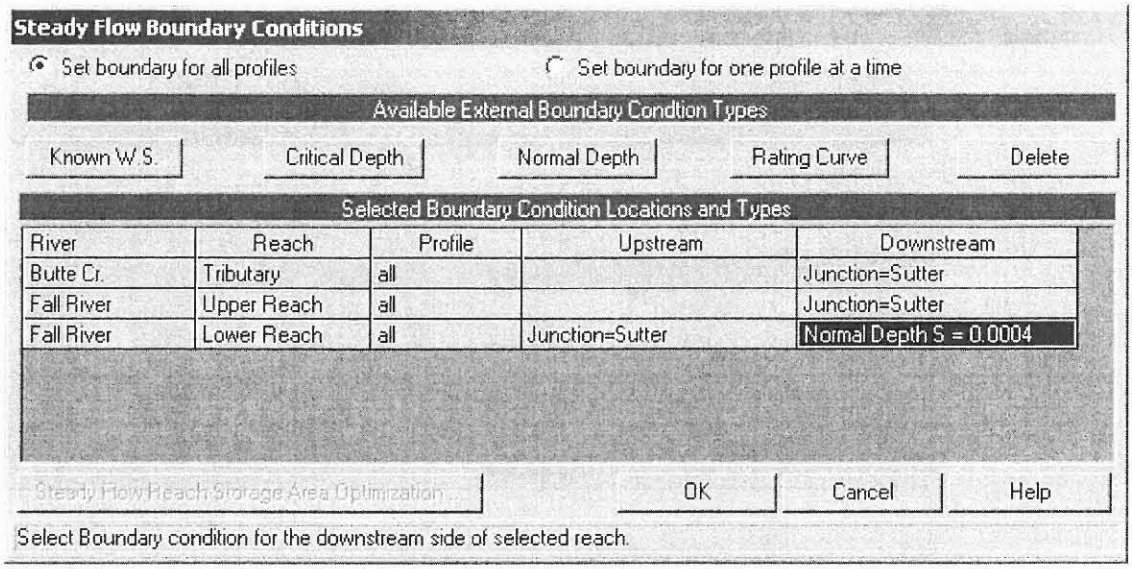


Figure 4.9 Steady Flow Boundary Conditions

The boundary conditions editor contains a table listing every river and 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 is connected in the geometric data editor. The user is only required to enter the necessary external boundary conditions.

In this example, it is assumed that the flow is subcritical throughout the river system. Therefore, it is only necessary to enter a boundary condition at the downstream end of the Fall River, Lower Reach. Boundary conditions are entered by first selecting the cell in which you wish to enter a boundary condition. Then the type of boundary condition is selected from the four available types listed above the table. The four types of boundary conditions are:

- Known water surface elevations
- Critical depth
- Normal depth
- Rating curve

For this example, use the normal depth boundary condition. Once you have selected the cell for the downstream end of Fall River, Lower Reach, press the **Normal Depth** button. A pop up box will appear requesting you to enter an average energy slope at the downstream end of the Fall River. Enter a value of 0.0004 (ft/ft) then press the **Enter** key. This completes all of the necessary boundary condition data. Press the **OK** button on the Boundary Conditions form to accept the data.

The last step in developing the steady flow data is to save the data to a file. To save the data, select the **Save Flow Data As** option from the **File** menu on the Steady Flow Data Editor. A pop up box will prompt you to enter a description of the flow data. For this example, enter "10, 2, and 1% chance events." Once the data are saved, you can close the Steady Flow Data Editor.

Performing the Hydraulic Calculations

Now that all of the data have been entered, we can calculate the steady water surface profiles. To perform the simulations, go to the HEC-RAS main window and select **Steady Flow Analysis** from the **Run** menu. The Steady Flow Analysis window should appear as shown in Figure 4.10, except yours will not have any plan titles yet.

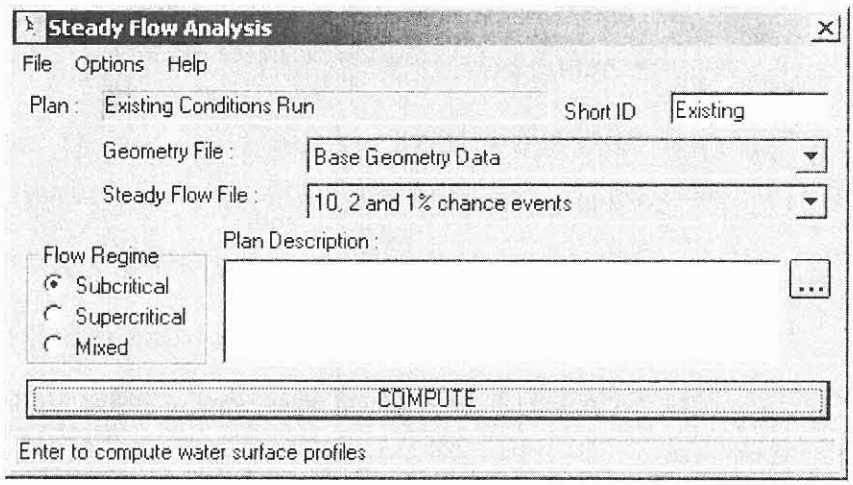


Figure 4.10 Steady Flow Analysis Simulation Window

The first step is to put together a **Plan**. The **Plan** defines which geometry and flow data are to be used, as well as providing a title and short identifier for the run. To establish a plan, select **New Plan** from the **File** menu on the Steady Flow Analysis window. Enter the plan title as "Existing Conditions Run" and then press the **OK** button. You will then be prompted to enter a short identifier. Enter a title of "Existing" in the **Short ID** box.

The next step is to select the desired flow regime for which the model will perform calculations. For this example we will be performing **Subcritical** flow calculations only. Make sure that **Subcritical** is the selected flow regime. Additional job control features are available from the **Options** menu bar, but none are required for this example. Once you have defined a plan and set all the desired job control information, the plan information should be saved. Saving the plan information is accomplished by selecting **Save Plan** from the **File** menu of the Steady Flow Analysis window.

Now that everything has been set, the steady flow computations can be performed by pressing the **Compute** button at the bottom of the Steady Flow Simulation window. Once the compute button has been pressed, a separate window will appear showing you the progress of the computations. Once the computations have been completed, the computation window can be closed by double clicking the upper left corner of the window. At this time the Steady Flow Simulation window can also be closed.

Viewing Results

Once the model has finished all of the computations successfully, you can begin viewing the results. Several output options are available from the **View** menu bar on the HEC-RAS main window. These options include:

- Cross section plots
- Profile plots
- General Profile Plot
- Rating curves
- X-Y-Z Perspective Plots
- Detailed tabular output at a specific cross section (cross section table)
- Limited tabular output at many cross sections (profile table)

Let's begin by plotting a cross section. Select **Cross Sections** from the **View** menu bar on the HEC-RAS main window. This will automatically bring up a plot of the first cross section in Butte Cr., as shown in Figure 4.11. Any cross section can be plotted by selecting the appropriate river, reach, and river station from the list boxes at the top of the cross section plot window. The user can also step through the plots by using the up and down arrow buttons. Several plotting features are available from the **Options** menu bar on the cross section plot window. These options include: zoom in; zoom out; selecting which plans, profiles and variables to plot; and control over lines, symbols, labels, scaling, and grid options.

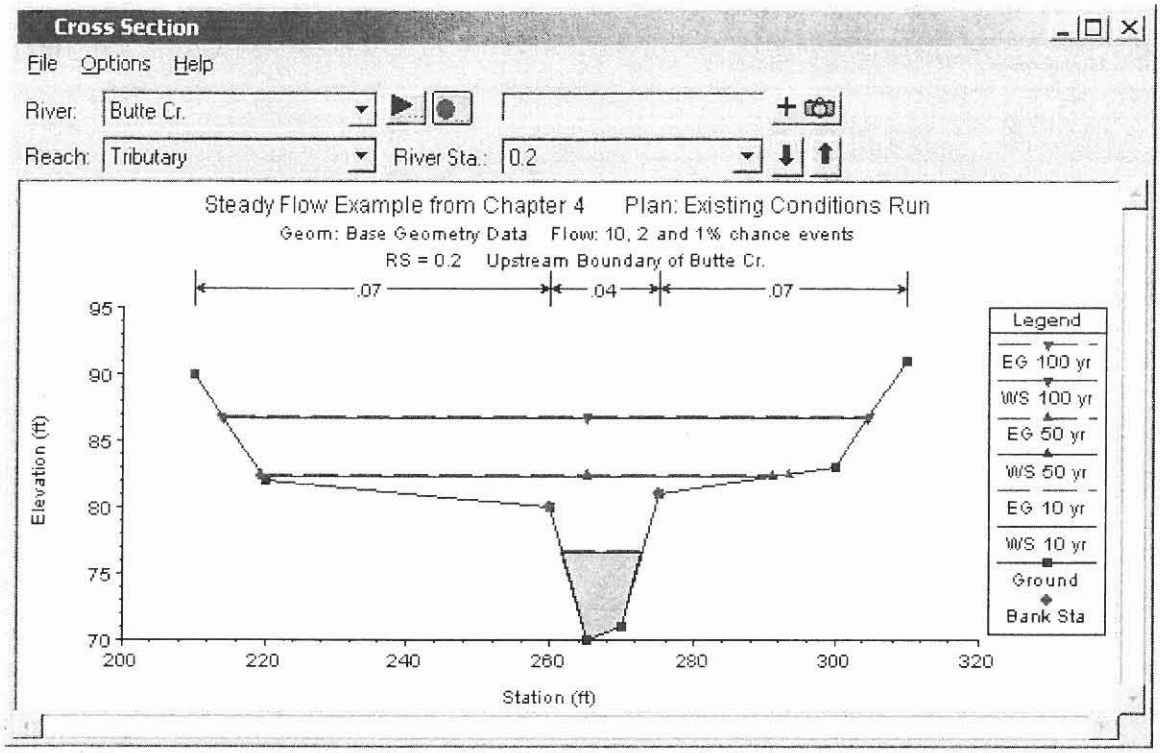


Figure 4.11 Cross Section Plot for Example Application

Select different cross sections to plot and practice using some of the features available under the **Options** menu bar.

Next let's plot a water surface profile. Select **Water Surface Profiles** from the **View** menu bar on the HEC-RAS main window. This will automatically bring up a water surface profile plot for the first reach, which is Butte Cr. in our example. To plot more than one reach, select **Reaches** from the **Options** menu bar on the profile plot. This option brings up a list of available rivers and reaches from which to choose. Select the Upper and Lower reaches of the Fall river. This should give you a profile plot as shown in Figure 4.12. Plot the additional profiles that were computed and practice using the other features available under the **Options** menu bar on the profile plot.

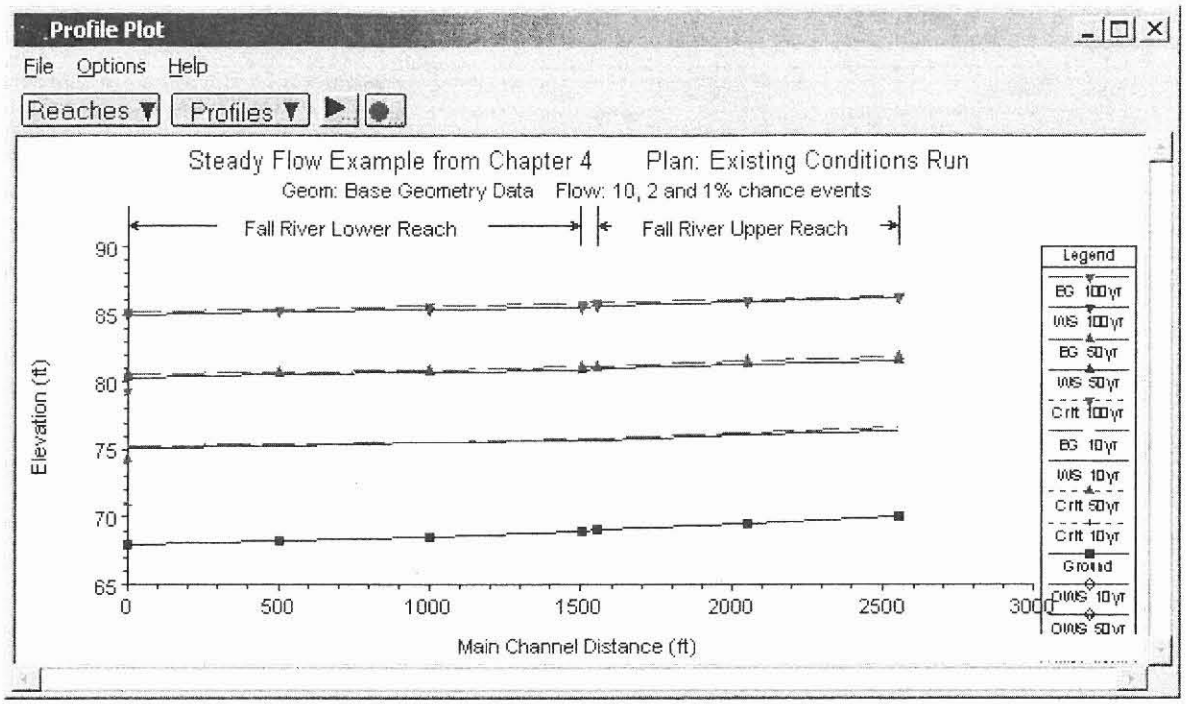


Figure 4.12 Profile Plot for Example Application

Now let's plot a computed rating curve. Select **Rating Curves** from the **View** menu on the HEC-RAS main window. A rating curve based on the computed water surface profiles will appear for the first cross section in Butte Cr., as shown in Figure 4.13. You can look at the computed rating curve for any location by selecting the appropriate river, reach, and river station from the list boxes at the top of the plot. Plotting options similar to the cross section and profile plots are available for the rating curve plots. Plot rating curves for various locations and practice using the available plotting options.

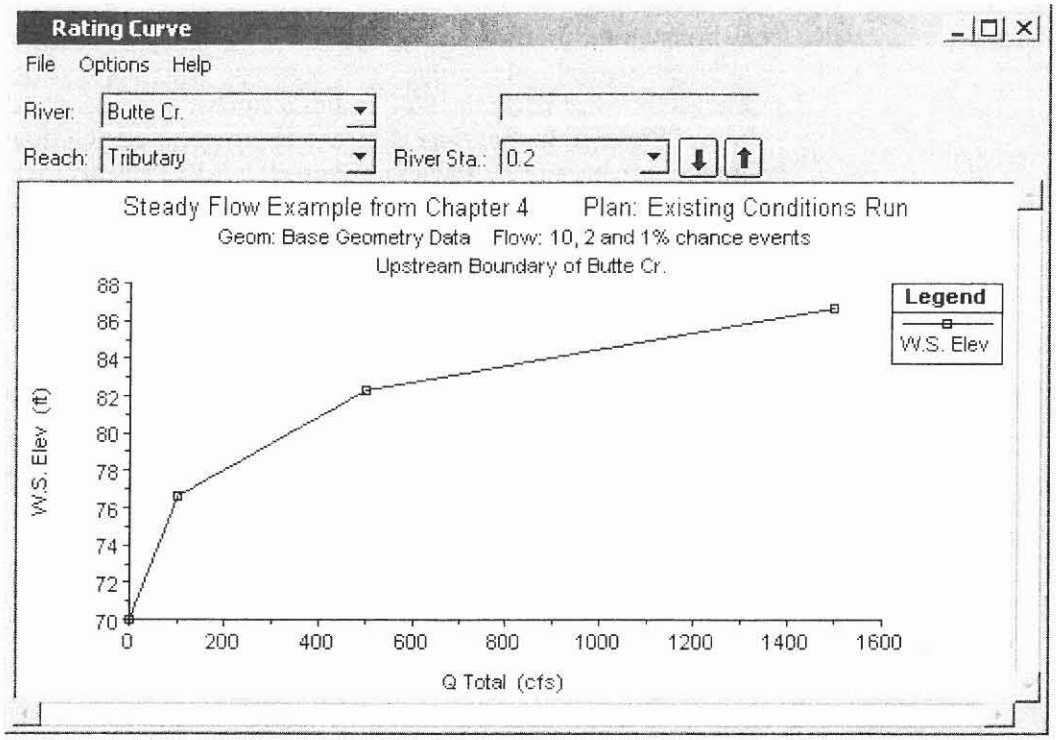


Figure 4.13 Computed Rating Curve for Example Application

Next look at an X-Y-Z Perspective Plot of the river system. From the **View** menu bar on the HEC-RAS main window, select **X-Y-Z Perspective Plots.** A multiple cross section perspective plot should appear on the screen. From the **Options** menu, select **Reaches**. A pop up window will appear allowing you to select which rivers and reaches you would like to have on the plot. Press the **Select All** button and then the **OK** button. Also, under the **Options** menu, select the **Profiles** option. Select profile two to be plotted from the three available profiles. Once you have selected these options, and X-Y-Z perspective plot should appear on the screen, similar to the one shown in Figure 4.14. Try rotating the perspective view in different directions, and select different reaches to look at.

Now let's look at some tabular output. Go to the **View** menu bar on the HEC-RAS main window. There are two types of tables available, a detailed output table and a profile summary table. Select **Detailed Output Tables** to get the first table to appear. The table should look like the one shown in Figure 4.15. This table shows detailed hydraulic information at a single cross section. Other cross sections can be viewed by selecting the appropriate reach and river mile from the table.

Now bring up the profile summary table. This table shows a limited number of hydraulic variables for several cross sections. There are several types of profile tables listed under the **Std. Tables** menu bar of the profile table window. Some of the tables are designed to provide specific information at hydraulic structures (e.g., bridges and culverts), while others provide generic information at all cross sections. An example of this type of table is shown in Figure 4.16.

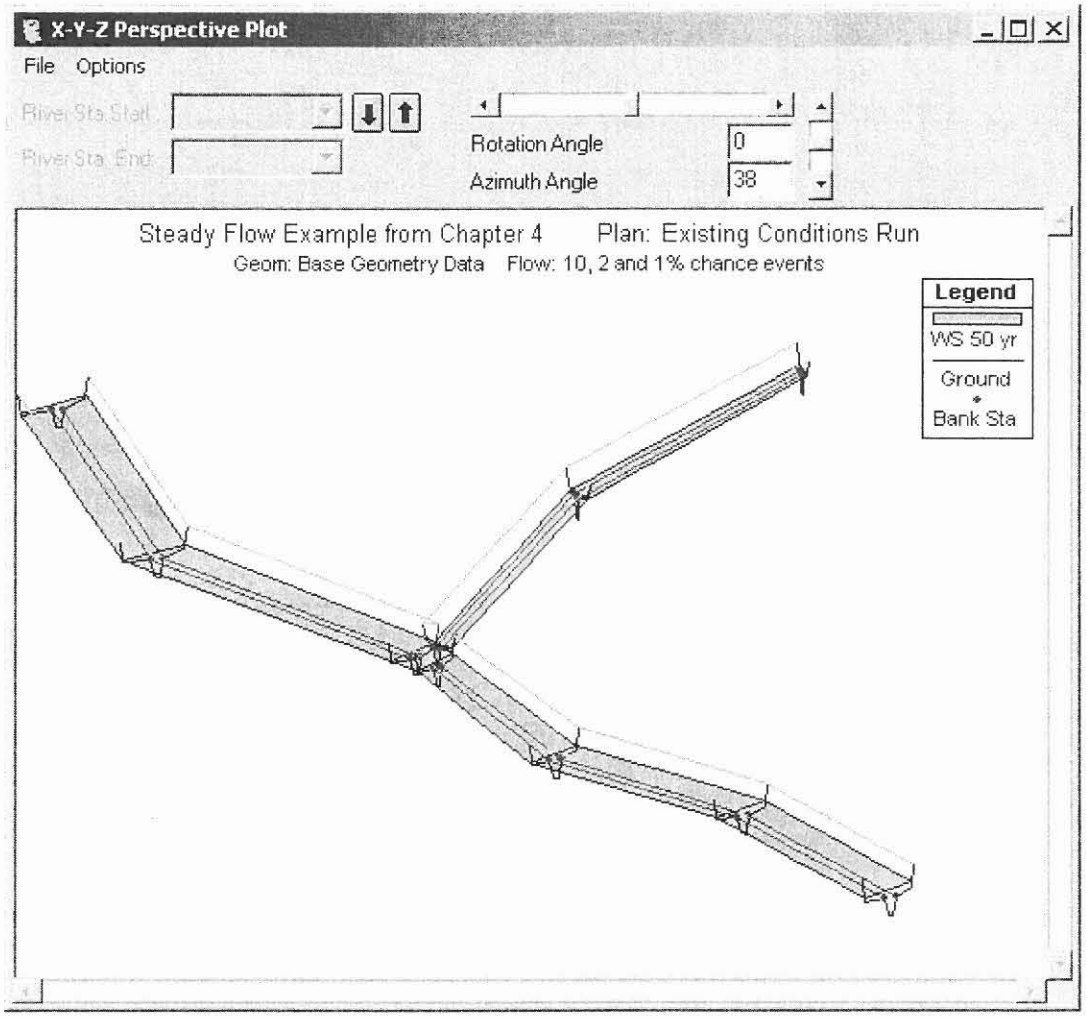


Figure 4.14 X-Y-Z Perspective Plot of All Three River Reaches

Cross Section Output

File Type Options Help

River: Fall River Profile: 50 yr

Reach Upper Reach Riv Sta: 10

Plan: Existing Fall River Upper Reach RS: 10 Profile: 50 yr

E.G. Elev (ft)	81.84	Element	Left OB	Channel	Right OB
Vel Head (ft)	0.23	Wt. n-Val.	0.060	0.035	0.050
W.S. Elev (ft)	81.61	Reach Len. (ft)	450.00	500.00	550.00
Crit W.S. (ft)		Flow Area (sq ft)	209.95	364.33	177.09
E.G. Slope (ft/ft)	0.000646	Area (sq ft)	209.95	364.33	177.09
Q Total (cfs)	2000.00	Flow (cfs)	246.75	1570.46	182.78
Top Width (ft)	232.22	Top Width (ft)	81.61	40.00	110.61
Vel Total (ft/s)	2.66	Avg. Vel. (ft/s)	1.18	4.31	1.03
Max Chl Dpth (ft)	11.61	Hydr. Depth (ft)	2.57	9.11	1.60
Conv. Total (cfs)	78681.0	Conv. (cfs)	9707.5	61782.8	7190.8
Length Wtd. (ft)	498.58	Wetted Per. (ft)	82.30	45.64	110.88
Min Ch El (ft)	70.00	Shear (lb/sq ft)	0.10	0.32	0.06
Alpha	2.10	Stream Power (lb/ft s)	0.12	1.39	0.07
Frctn Loss (ft)	0.31	Cum Volume (acre-ft)	4.25	8.98	4.49
C & E Loss (ft)	0.00	Cum SA (acres)	1.49	0.92	2.46

Errors, Warnings and Notes

Conveyance weighted Manning's n for the left overbank.

Figure 4.15 Detailed Tabular Output at a Cross Section

Profile Output Table - Standard Table 1

File Options Std. Tables User Tables Locations Help

HEC-RAS Plan: Existing Profile: 50 yr (Reload Data)

River	Reach	River Sta	Q Total	Min Ch El	W.S. Elev	Crit W.S.	E.G. Elev	E.G. Slope	Vel Chnl	Flow Area	Top Width
			(cfs)	(ft)	(ft)	(ft)	(ft)	(ft/ft)	(ft/s)	(sq ft)	(ft)
Fall River	Upper Reach	10	2000.00	70.00	81.61		81.84	0.000646	4.31	751.24	232.22
Fall River	Upper Reach	9.9	2000.00	69.50	81.31		81.53	0.000599	4.21	754.61	213.35
Fall River	Upper Reach	9.8	2000.00	69.10	80.97		81.22	0.000630	4.34	690.50	178.78
Fall River	Lower Reach	9.79	2500.00	69.00	80.92		81.18	0.000640	4.49	839.21	214.62
Fall River	Lower Reach	9.7	2500.00	68.50	80.76		80.92	0.000360	3.51	1094.31	258.24
Fall River	Lower Reach	9.6	2700.00	68.20	80.55		80.72	0.000400	3.73	1117.55	258.43
Fall River	Lower Reach	9.5	2700.00	68.00	80.35	74.20	80.52	0.000400	3.73	1117.53	258.43
Butte Cr.	Tributary	0.2	500.00	70.00	82.30		82.48	0.001176	3.54	189.77	71.65
Butte Cr.	Tributary	0.1	500.00	69.40	81.72		81.89	0.001164	3.52	190.91	71.87
Butte Cr.	Tributary	0.0	500.00	69.10	81.01		81.23	0.001495	3.86	162.99	64.67

Figure 4.16 Tabular Output in Profile Format

Printing Graphics and Tables

All of the plots and tables can be sent directly to a printer/plotter or passed through the Windows clipboard to another program (e.g., a word processor). The printer or plotter that gets used is based on what you currently have selected as the default printer for Windows. The user has the ability to change many of the default printer settings (e.g., portrait to landscape) before printing occurs.

Sending Graphics Directly to the Printer

To send a graphic to the printer/plotter, do the following:

1. Display the graphic of interest (cross section, profile, rating curve, or river system schematic) on the screen.

2. Using the available options (scaling, labels, grid, etc.), modify the plot to be what you would like printed out.

3. Select **Print** from the **File** menu of the displayed graphic. Once Print is selected, a **Printer Options** window will appear, giving the user the opportunity to change any of the default printer settings. Once you have the print settings the way you want them, press the **Print** button on the **Printer Options** window and the plot will automatically be sent to the Windows Print Manager. From that point the Windows Print Manager will control the printing.

Sending Graphics to the Windows Clipboard

To pass a graphic to the Windows clipboard and then to another program, do the following:

1. Display the graphic of interest on the screen.

2. Using the available options, modify the plot to be what you want it to look like.

3. Select **Copy to Clipboard** from the **File** menu of the displayed graphic. The plot will automatically be sent to the Windows clipboard.

4. Bring up the program that you want to pass the graphic into (e.g., word processor). Select **Paste** from the **Edit** menu of the receiving program. Once the graphic is pasted in, it can be resized to the desired dimensions.

Sending Tables Directly to the Printer

To send a table to the printer, do the following:

1. Bring up the desired table from the tabular output section of the program.

2. Select **Print** from the **File** menu of the displayed table. Once the Print option is selected, a **Printer Options** window will appear. Set any print options that are desired then press the **Print** button. This will send the entire table to the Windows Print Manager. From this point the Windows Print Manager will control the printing of the table.

The profile type of table allows you to print a specific portion of the table, rather than the whole thing. If you desire to only print a portion of the table, do the following:

1. Display the desired profile type table on the screen.

2. Using the mouse, press down on the left mouse button and highlight the area of the table that you would like to print. To get an entire row or column, press down on the left mouse button while moving the pointer across the desired row or column headings.

3. Select **Print** from the **File** menu of the displayed table. Only the highlighted portion of the table and the row and column headings will be sent to the Windows Print Manager.

Sending Tables to the Windows Clipboard

To pass a table to the Windows clipboard and then to another program, do the following:

1. Display the desired table on the screen.

2. Select **Copy to Clipboard** from the **File** menu of the displayed table.

3. Bring up the program that you want to pass the table into. Select **Paste** from the **Edit** menu of the receiving program.

Portions of the profile table can be sent to the clipboard in the same manner as sending them to the printer.

Practice sending graphics and tables to the printer and the clipboard with the example data set that you currently have open.

Exiting the Program

Before you exit the HEC-RAS software, make sure you have saved all the data. This can be accomplished easily by selecting **Save Project** from the **File** menu on the HEC-RAS main window. Any data (geometric, flow, and plan data) that have not been saved will automatically be saved for you.

To exit the HEC-RAS software, select **Exit** from the **File** menu of the HEC-RAS main window. The program will prompt you to save the project if the data have not been saved previously.

CHAPTER 5

Working With Projects

To create a river hydraulics application with HEC-RAS, you work with projects. A **project** is a collection of files that are used to build a model. This chapter describes projects and how you build and manage them.

Contents

- Understanding Projects
- Elements of a Project
- Creating, Opening, Saving, Renaming, and Deleting Projects
- Project Options

Understanding Projects

As you develop an application, the management of all the files that get created is accomplished through the user interface. When a new project is started, the user is requested to enter a title and filename for the project. All other data are automatically stored by the user interface using the same name as the project file, except for the three character extension. A project consists of:

- One **Project** file (.PRJ)
- One file for each **Plan** (.P01 to .P99)
- One **Run** file for each steady flow plan (.R01 to .R99)
- One **Output** file for each plan (.O01 to .O99)
- One file for each set of **Geometry** data (.G01 to .G99)
- One file for each set of **Steady Flow** data (.F01 to .F99)
- One file for each set of **Unsteady Flow** data (.U01 to .U99)
- One file for each set of **Sediment** data (.S01 to .S99)
- One file for each set of **Hydraulic Design** data (.H01 to .H99)

The **Project File** contains: the title of the project; the units system of the project; a list of all the files that are associated with the project; and a list of default variables that can be set from the interface. Also included in the project file is a reference to the last plan that the user was working with. This information is updated every time you save the project.

Elements of a Project

The following sections describe the various types of files that can be included in a project. All of these files are either created by the user interface or the various computation engines. The modeler interacts with the data through the user interface, and is not required to create or edit any of these files directly.

Plan Files

Plan files have the extension .P01 to .P99. The "P" indicates a Plan file, while the number represents the plan number. As plans are created, they are numbered from 01 to 99. The plan file contains: a description and short identifier for the plan; a list of files that are associated with the plan (e.g., geometry file and steady flow file); and a description of all the simulation options that were set for the plan. The plan file is created automatically by the interface each time the user selects **New Plan** or **Save Plan As** from the simulation windows.

Run Files

Run files have the extension .R01 to .R99. The "R" indicates a Run file, while the number represents an association to a particular plan file. A file with an extension of .R01 is the run file that corresponds to the plan file with the extension .P01. The run file contains all of the necessary data to perform the computations that are requested by the associated plan file. For example, if a steady flow analysis is requested, the run file will contain geometry data, steady flow data, and all the necessary computational options that are associated with the plan file. The run file contains the input to any of the computational engines available in the HEC-RAS system. The run file is automatically generated by the interface whenever the user presses the **Compute** button on the Simulation windows. The run file is in an ASCII format, but it is not self explanatory.

Output Files

Output files have the extension .O01 to .O99. The "O" indicates an Output file, while the number represents an association to a particular plan file. A file with the extension .O12 is the output file that corresponds to the plan file with an extension .P12. The output file contains all of the computed results from the requested computational engine. For example, if a steady flow analysis is requested, the output file will contain results from the steady flow computational engine. The output files are in a binary file format and can only be read from the user interface.

Geometry Files

Geometry files have the extension .G01 to .G99. The "G" indicates a Geometry file, while the number corresponds to the order in which they were saved for that particular project. Geometry files contain all of the geometric data for the river system being analyzed. The geometric data consist of: cross section information; hydraulic structures data (e.g., bridges and culverts); coefficients; and modeling approach information. The geometry data are stored in an ASCII format. The file contains key words to describe each piece of data, and is for-the-most-part self explanatory. A geometry file is created by the user interface whenever the modeler selects **New Geometry Data** or **Save Geometry Data As** from the Geometric Data window.

Steady Flow Data Files

Steady flow data files have the extension .F01 to .F99. The "F" represents that it is a steady Flow data file, while the number corresponds to the order in which they were saved for that particular project. Steady flow data files contain: the number of profiles to be computed; flow data; and boundary conditions for each reach. The steady flow data files are stored in an ASCII format. The file contains key words to describe each piece of data, and is for-the-most-part self explanatory. Steady flow data files are automatically created by the user interface when the modeler selects **New Flow Data** or **Save Flow Data As** from the Steady Flow Data window.

Unsteady Flow Data Files

Unsteady flow data files have the extension .U01 to .U99. The "U" represents that it is an Unsteady flow data file, while the number corresponds to the order in which they were saved for that particular project. Unsteady flow data files contain: flow hydrographs at the upstream boundaries; starting flow conditions; and downstream boundary conditions. The unsteady flow data files are stored in an ASCII format. The file contains key words to describe each piece of data, and is for-the-most-part self explanatory. Unsteady flow data files are automatically created by the user interface when the modeler selects **New Flow Data** or **Save Flow Data As** from the Unsteady Flow Data window.

Sediment Data Files

Sediment data files have the extension .S01 to .S99. The "S" represents that it is a Sediment data file, while the number corresponds to the order in which they were saved for that particular project. Sediment data files contain: flow data; boundary conditions for each reach; and sediment data. The sediment data files are stored in an ASCII format. The file contains key words to describe each piece of data, and is for-the- most-part self explanatory. Sediment data files are automatically created by the user interface when the modeler selects **New Sediment Data** or **Save Sediment Data As** from the Sediment Data window. Currently, the sediment option is not available in HEC-RAS. This option will be included in a future version.

Hydraulic Design Data Files

Hydraulic design data files have the extension .H01 to .H99. The "H" represents that it is a Hydraulic design data file, while the number corresponds to the order in which they were saved for that particular project. Hydraulic design data files contain information corresponding to the type of hydraulic design calculation that is requested. The Hydraulic design data files are stored in an ASCII format. The file contains key words to describe each piece of data, and is for-the most-part self explanatory. Hydraulic Design data files are automatically created by the user interface when the modeler selects **New Hydraulic Design Data** or **Save Hydraulic Design Data As** from the **File** menu of the Hydraulic Design Functions window.

A schematic diagram of how the data files fit together is shown in Figure 5.1 on the next page. In this example there are three plans in the project. Each plan represents a specific set of steady flow data and geometry data. In this example there are three geometry files and one steady flow file. The first geometry file could represent the existing conditions of the stream. The second and third geometry file could represent some modification of that base geometry file, such as adding a bridge or culvert crossing; a channel modification; different roughness coefficients; or any other change to the base geometry file. A plan is formulated by selecting a steady flow file and a geometry file, and then saving that plan with a specific title and short identifier. For more information about formulating plans, see Chapter 7 of the HEC-RAS User's Manual and Chapter 7 of the HEC-RAS Applications Guide.

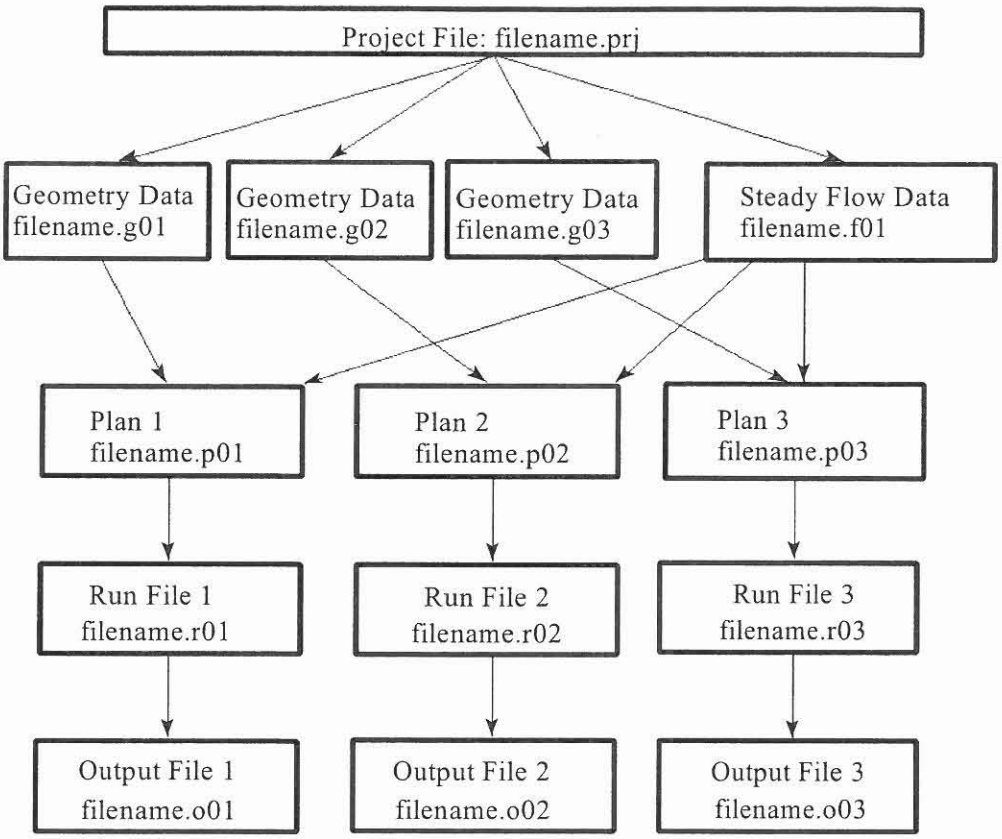


Figure 5.1 Schematic of Project Data Files.

Creating, Opening, Saving, Renaming, and Deleting Projects

The following commands from the **File** menu of the HEC-RAS main window allow you to create, open, save, rename, and delete projects.

File menu command	Description
New Project	Closes the current project, prompting you to save the data if anything has been changed. The user is then prompted to enter a title and filename for the new project.
Open Project	Closes the current project, prompting you to save the data if anything has been changed. Opens an existing project and all of the associated files.
Save Project	Updates the project file and all other files in which data have been modified.
Save Project As	Updates the project file and all other associated data, saving all the information to a new filename that you specify.
Rename Project	Allows the user to rename the title of the currently opened project.
Delete Project	Deletes the project file and all other files associated with the selected project. The user is prompted to make sure that they really want to delete all of the files.

These commands are the same for all of the other data types that get created by the user interface (Plan data, geometry data, steady flow data, unsteady flow data, sediment data, and hydraulic design data).

Project Options

From the **Options** menu of the main HEC-RAS window, the user can set several default project options. These options include: setting default margins and color control for printing; setting default hydraulic variables; establishing the default units system (English or Metric); and converting existing projects to a different units system (English to Metric or Metric to English). The following four options are available from the **Options** menu:

Options menu command	Description
Program Setup	
- Default File Viewer	This option allows the user to change which program is used for viewing the report generator and logfile output. The default is the Windows Write program. The user can change this to any file viewer on their system.
- AVI Bin Directory	This option allows the user to set the default directory for where HEC-RAS will store bitmap files that it makes when creating an AVI movie file.
- Open last project	When this option is selected, the program will automatically open the last project worked on, during startup.
- Automatically backup data	When this option is checked, the program will automatically make a backup of the currently opened project, plan, geometry, and flow files. The backup files are updated at specific timed intervals, which is user controlled. The backup files are stored in the \HEC\RAS directory, with the titles RasBackup.prj, RasBackup.p01, RasBackup.g01, and RasBackup.f01.
- Set time for automatic backup	This option allows the user to control the time interval between updating the backup files. The default value is 20 minutes.

Options menu command	Description
Default Parameters	This option allows the user to set defaults for some of the hydraulic variables.
Unit System	This option allows the user to set the default units system to either English or Metric. Once the units system is set, the program assumes that all input data are entered in that units system. Likewise, the display of all output data will be done in the default units system.
Convert Project Units	This option allows the user to convert an existing project from one units system to another. Projects can be converted from English to Metric or from Metric to English.

CHAPTER 5

Working With Projects

To create a river hydraulics application with HEC-RAS, you work with projects. A **project** is a collection of files that are used to build a model. This chapter describes projects and how you build and manage them.

Contents

- Understanding Projects
- Elements of a Project
- Creating, Opening, Saving, Renaming, and Deleting Projects
- Project Options

Understanding Projects

As you develop an application, the management of all the files that get created is accomplished through the user interface. When a new project is started, the user is requested to enter a title and filename for the project. All other data are automatically stored by the user interface using the same name as the project file, except for the three character extension. A project consists of:

- One **Project** file (.PRJ)
- One file for each **Plan** (.P01 to .P99)
- One **Run** file for each steady flow plan (.R01 to .R99)
- One **Output** file for each plan (.O01 to .O99)
- One file for each set of **Geometry** data (.G01 to .G99)
- One file for each set of **Steady Flow** data (.F01 to .F99)
- One file for each set of **Unsteady Flow** data (.U01 to .U99)
- One file for each set of **Sediment** data (.S01 to .S99)
- One file for each set of **Hydraulic Design** data (.H01 to .H99)

The **Project File** contains: the title of the project; the units system of the project; a list of all the files that are associated with the project; and a list of default variables that can be set from the interface. Also included in the project file is a reference to the last plan that the user was working with. This information is updated every time you save the project.

Elements of a Project

The following sections describe the various types of files that can be included in a project. All of these files are either created by the user interface or the various computation engines. The modeler interacts with the data through the user interface, and is not required to create or edit any of these files directly.

Plan Files

Plan files have the extension .P01 to .P99. The "P" indicates a Plan file, while the number represents the plan number. As plans are created, they are numbered from 01 to 99. The plan file contains: a description and short identifier for the plan; a list of files that are associated with the plan (e.g., geometry file and steady flow file); and a description of all the simulation options that were set for the plan. The plan file is created automatically by the interface each time the user selects **New Plan** or **Save Plan As** from the simulation windows.

Run Files

Run files have the extension .R01 to .R99. The "R" indicates a Run file, while the number represents an association to a particular plan file. A file with an extension of .R01 is the run file that corresponds to the plan file with the extension .P01. The run file contains all of the necessary data to perform the computations that are requested by the associated plan file. For example, if a steady flow analysis is requested, the run file will contain geometry data, steady flow data, and all the necessary computational options that are associated with the plan file. The run file contains the input to any of the computational engines available in the HEC-RAS system. The run file is automatically generated by the interface whenever the user presses the **Compute** button on the Simulation windows. The run file is in an ASCII format, but it is not self explanatory.

Output Files

Output files have the extension .O01 to .O99. The "O" indicates an Output file, while the number represents an association to a particular plan file. A file with the extension .O12 is the output file that corresponds to the plan file with an extension .P12. The output file contains all of the computed results from the requested computational engine. For example, if a steady flow analysis is requested, the output file will contain results from the steady flow computational engine. The output files are in a binary file format and can only be read from the user interface.

Geometry Files

Geometry files have the extension .G01 to .G99. The "G" indicates a Geometry file, while the number corresponds to the order in which they were saved for that particular project. Geometry files contain all of the geometric data for the river system being analyzed. The geometric data consist of: cross section information; hydraulic structures data (e.g., bridges and culverts); coefficients; and modeling approach information. The geometry data are stored in an ASCII format. The file contains key words to describe each piece of data, and is for-the-most-part self explanatory. A geometry file is created by the user interface whenever the modeler selects **New Geometry Data** or **Save Geometry Data As** from the Geometric Data window.

Steady Flow Data Files

Steady flow data files have the extension .F01 to .F99. The "F" represents that it is a steady Flow data file, while the number corresponds to the order in which they were saved for that particular project. Steady flow data files contain: the number of profiles to be computed; flow data; and boundary conditions for each reach. The steady flow data files are stored in an ASCII format. The file contains key words to describe each piece of data, and is for-the-most-part self explanatory. Steady flow data files are automatically created by the user interface when the modeler selects **New Flow Data** or **Save Flow Data As** from the Steady Flow Data window.

Unsteady Flow Data Files

Unsteady flow data files have the extension .U01 to .U99. The "U" represents that it is an Unsteady flow data file, while the number corresponds to the order in which they were saved for that particular project. Unsteady flow data files contain: flow hydrographs at the upstream boundaries; starting flow conditions; and downstream boundary conditions. The unsteady flow data files are stored in an ASCII format. The file contains key words to describe each piece of data, and is for-the-most-part self explanatory. Unsteady flow data files are automatically created by the user interface when the modeler selects **New Flow Data** or **Save Flow Data As** from the Unsteady Flow Data window.

Sediment Data Files

Sediment data files have the extension .S01 to .S99. The "S" represents that it is a Sediment data file, while the number corresponds to the order in which they were saved for that particular project. Sediment data files contain: flow data; boundary conditions for each reach; and sediment data. The sediment data files are stored in an ASCII format. The file contains key words to describe each piece of data, and is for-the- most-part self explanatory. Sediment data files are automatically created by the user interface when the modeler selects **New Sediment Data** or **Save Sediment Data As** from the Sediment Data window. Currently, the sediment option is not available in HEC-RAS. This option will be included in a future version.

Hydraulic Design Data Files

Hydraulic design data files have the extension .H01 to .H99. The "H" represents that it is a Hydraulic design data file, while the number corresponds to the order in which they were saved for that particular project. Hydraulic design data files contain information corresponding to the type of hydraulic design calculation that is requested. The Hydraulic design data files are stored in an ASCII format. The file contains key words to describe each piece of data, and is for-the most-part self explanatory. Hydraulic Design data files are automatically created by the user interface when the modeler selects **New Hydraulic Design Data** or **Save Hydraulic Design Data As** from the **File** menu of the Hydraulic Design Functions window.

A schematic diagram of how the data files fit together is shown in Figure 5.1 on the next page. In this example there are three plans in the project. Each plan represents a specific set of steady flow data and geometry data. In this example there are three geometry files and one steady flow file. The first geometry file could represent the existing conditions of the stream. The second and third geometry file could represent some modification of that base geometry file, such as adding a bridge or culvert crossing; a channel modification; different roughness coefficients; or any other change to the base geometry file. A plan is formulated by selecting a steady flow file and a geometry file, and then saving that plan with a specific title and short identifier. For more information about formulating plans, see Chapter 7 of the HEC-RAS User's Manual and Chapter 7 of the HEC-RAS Applications Guide.

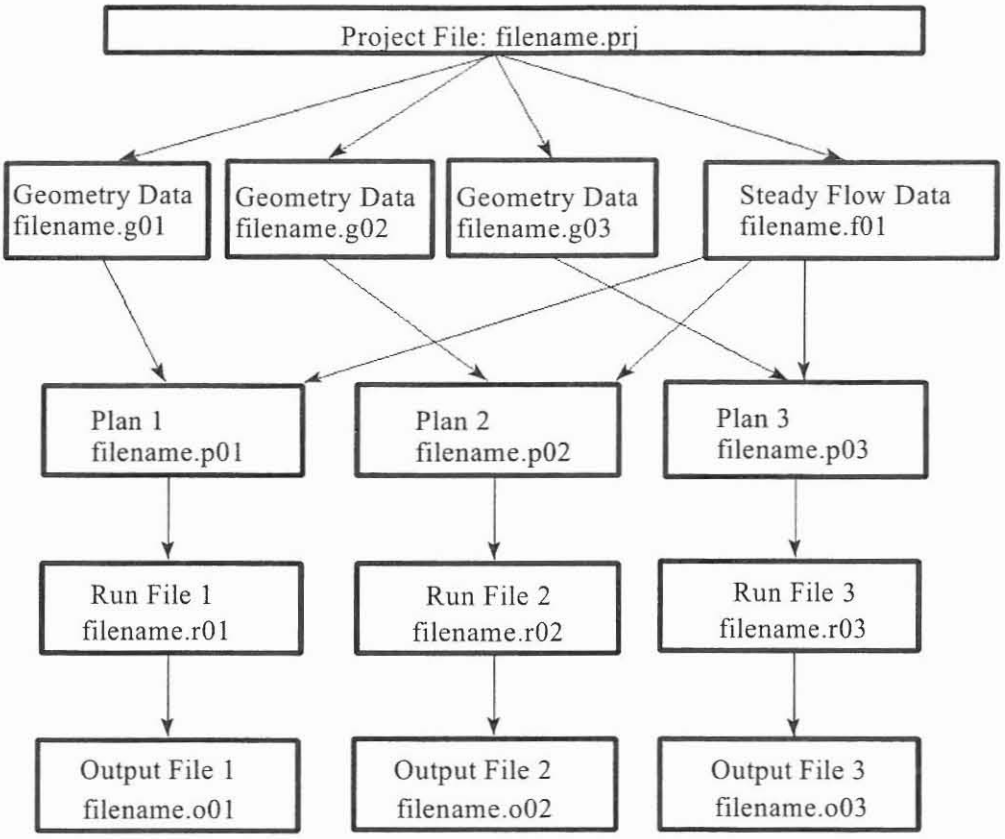


Figure 5.1 Schematic of Project Data Files.

Creating, Opening, Saving, Renaming, and Deleting Projects

The following commands from the **File** menu of the HEC-RAS main window allow you to create, open, save, rename, and delete projects.

File menu command	Description
New Project	Closes the current project, prompting you to save the data if anything has been changed. The user is then prompted to enter a title and filename for the new project.
Open Project	Closes the current project, prompting you to save the data if anything has been changed. Opens an existing project and all of the associated files.
Save Project	Updates the project file and all other files in which data have been modified.
Save Project As	Updates the project file and all other associated data, saving all the information to a new filename that you specify.
Rename Project	Allows the user to rename the title of the currently opened project.
Delete Project	Deletes the project file and all other files associated with the selected project. The user is prompted to make sure that they really want to delete all of the files.

These commands are the same for all of the other data types that get created by the user interface (Plan data, geometry data, steady flow data, unsteady flow data, sediment data, and hydraulic design data).

Project Options

From the **Options** menu of the main HEC-RAS window, the user can set several default project options. These options include: setting default margins and color control for printing; setting default hydraulic variables; establishing the default units system (English or Metric); and converting existing projects to a different units system (English to Metric or Metric to English). The following four options are available from the **Options** menu:

Options menu command	Description
Program Setup	
- Default File Viewer	This option allows the user to change which program is used for viewing the report generator and logfile output. The default is the Windows Write program. The user can change this to any file viewer on their system.
- AVI Bin Directory	This option allows the user to set the default directory for where HEC-RAS will store bitmap files that it makes when creating an AVI movie file.
- Open last project	When this option is selected, the program will automatically open the last project worked on, during startup.
- Automatically backup data	When this option is checked, the program will automatically make a backup of the currently opened project, plan, geometry, and flow files. The backup files are updated at specific timed intervals, which is user controlled. The backup files are stored in the \HEC\RAS directory, with the titles RasBackup.prj, RasBackup.p01, RasBackup.g01, and RasBackup.f01.
- Set time for automatic backup	This option allows the user to control the time interval between updating the backup files. The default value is 20 minutes.

Options menu command	Description
Default Parameters	This option allows the user to set defaults for some of the hydraulic variables.
Unit System	This option allows the user to set the default units system to either English or Metric. Once the units system is set, the program assumes that all input data are entered in that units system. Likewise, the display of all output data will be done in the default units system.
Convert Project Units	This option allows the user to convert an existing project from one units system to another. Projects can be converted from English to Metric or from Metric to English.

CHAPTER 6

Entering and Editing Geometric Data

Geometric data consist of establishing the connectivity of the river system (River System Schematic), entering cross-section data, defining all the necessary junction information, adding hydraulic structure data (bridges, culverts, weirs, etc.) and cross section interpolation. The geometric data is entered by selecting **Geometric Data** from the **Edit** menu on the HEC-RAS main window. Once this option is selected, the Geometric Data window will appear as shown in Figure 6.1. The drawing area will be blank on your screen, until you have drawn in your own river system schematic.

This chapter describes how to enter and edit all of the necessary geometric data for a river system.

Contents

- Developing the River System Schematic
- Cross Section Data
- Stream Junctions
- Bridges and Culverts
- Multiple Bridge and/or Culvert Openings
- Inline Structures (Weirs and Gated Spillways)
- Lateral Structures (Weirs, Gated Spillways, Culverts, and Rating Curves)
- Storage Areas
- Storage Area Connections
- Pump Stations
- Cross Section Interpolation
- River Ice
- Viewing and Editing Data Through Tables
- Importing Geometric Data
- Geometric Data Tools
- Attaching and Viewing Pictures
- Saving the Geometric Data

Developing the River System Schematic

Building The Schematic

The modeler develops the geometric data by first drawing in the river system schematic on the Geometric Data window (Figure 6.1). The River System Schematic is a diagram of how the stream system is connected together. The river system is drawn on a reach-by-reach basis, by pressing the **River Reach** button and then drawing in a reach from upstream to downstream (in the positive flow direction). Each reach is identified with a **River Name** and a **Reach Name**. The River Name should be the actual name of the stream, while the reach name is an additional qualifier for each hydraulic reach within that river. A river can be comprised of one or more reaches. Reaches start or end at locations where two or more streams join together or spilt apart. Reaches can also start or end at the open ends of the river system being modeled. In other words, the first and last cross section that will be modeled in a particular river will be the start or end of a particular reach.

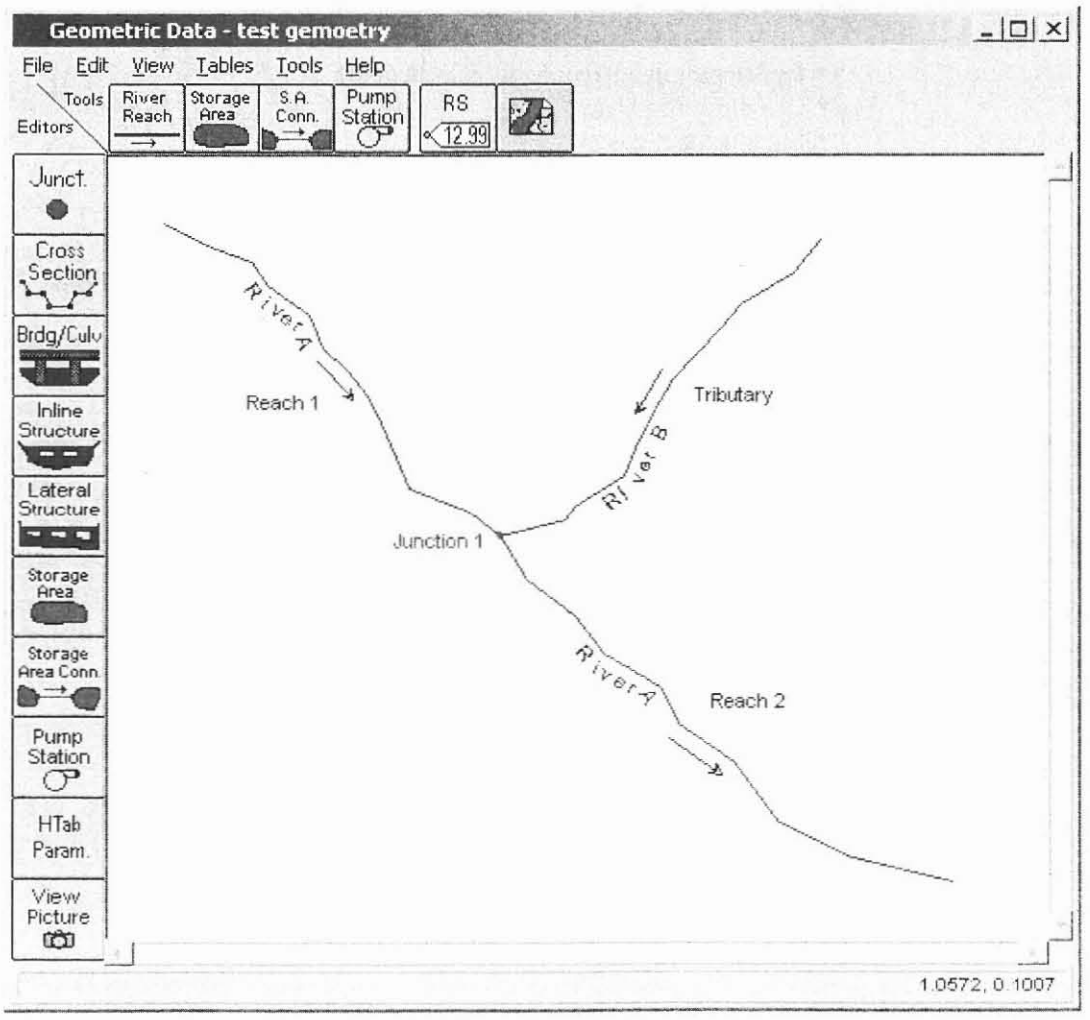


Figure 6.1 Geometric Data Editor Window

Reaches are drawn as multi-segmented lines. Each reach must have at least two points, defining the start and end of the reach. However, it is more typical to draw a reach with several points that would follow along the main channel invert of the stream. To draw a reach, first press the **River Reach** button at the top of the Geometric Data editor, on the tools button bar. Move the mouse pointer to the location on the drawing area that you would like to have the reach begin (upstream end of the reach). Click the left mouse button once to define the first point of the reach. Move the mouse and continue to click the left mouse button to add additional points to the reach. To end a reach, move the mouse pointer to the location in which you would like the last point of the reach to be located, then double click the left mouse button. After the reach is drawn, the user is prompted to enter the **River Name** and the **Reach Name** to identify the reach. The river and reach identifiers are limited to sixteen characters in length. If a particular River Name has already been entered for a previously defined reach of the same river, the user should simply select that river name from the list of available rivers in the river name text box. As reaches are connected together, junctions are automatically formed by the interface. The modeler is also prompted to enter an identifier for each junction. Junctions are locations where two or more streams join together or split apart. Junction identifiers are also limited to sixteen characters. An example of a simple stream system schematic is shown in Figure 6.1.

In addition to river reaches, the user can draw **Storage Areas, Storage Area Connections,** and **Pump Stations**. A storage area is used to define an area in which water can flow into and out of. The water surface in a storage area is assumed to be a level pool. Storage areas can be connected to river reaches as well as other storage areas. The user connects a storage area to a reach by using the lateral structure option. Storage areas can be connected to other storage areas by using a storage area connection between the storage areas. Storage area connections consist of culverts, gated spillways and a weir. The user can set up a storage area connection as just a weir, a weir and culverts, or a weir and gated spillways. For connections between a storage area and a river reach, the lateral structure option is the only means of doing this in HEC-RAS.

To draw a **Storage Area**, select the storage area button at the top of the geometric editor window. Storage areas are drawn as polygons. Move the mouse pointer to the location in which you would like to start drawing the storage area. Press the left mouse button one time to start adding points to define the storage area. Continue using single left mouse clicks to define the points of the storage area. To end the storage area use a double left mouse click. The storage area will automatically be closed into a polygon. Once you have finished drawing the storage area, a window will appear asking you to enter a name for the storage area. To enter and edit the data for a storage area, use the storage area editor button on the left panel of the geometric data window.

To enter a **Storage Area Connection,** select the storage area connection button at the top of the Geometric data editor. Move the mouse pointer to the storage area that you want to connect from. Then click the left mouse pointer one time to start the drawing of the connection. You can continue to use single mouse clicks to add as many points as you want into the line that represents the storage area connection. When you want to end the connection, place the mouse pointer over the storage area that you want to connect to, and then double click the mouse pointer. A window will pop up asking you to enter a name for the storage area connection. The direction in which you draw the storage area connection is important for establishing the positive flow direction for the flow. If you want the program to output positive flow when the flow is going from one storage area to another storage area, then you must draw the connection in that order. This is establishing the positive flow direction for the storage area connection. If flow happens to go in the other direction during the calculations, that flow will be output as negative numbers. To enter and edit the data for a storage area connection use the **Storage Area Connection** data editor on the left panel of the geometric data window.

Pump stations can be connected between two storage areas, between a storage area and a river reach, or between two river reaches. To add a pump station to the schematic, click the **Pump Station** button at the top, under the tools button bar, and then left click on the schematic at the location where you want to place the pump station. To connect the pump station, either left click over top of the pump station and select edit, or just click on the pump station editor from the edit tool bar. Connecting pumps is accomplished by picking from and to locations from the pump data editor.

Adding Tributaries into an Existing Reach

If you would like to add a tributary or bifurcation into the middle of an existing reach, this can be accomplished by simply drawing the new reach, and connecting it graphically to the existing reach at the location where you would like the new junction to be formed. Once the new reach is connected into the middle of an existing reach, you will first be prompted to enter a River and Reach identifier for the new reach. After entering the river and reach identifiers, you will be asked if you want to "Split" the existing reach into two reaches. If you answer "yes", you will be prompted to enter a Reach identifier for the lower portion of the existing reach and a Junction name for the newly formed stream junction.

Editing The Schematic

There are several options available for editing the river system schematic. These options include: changing labels, moving objects (such as labels, junctions, and points in a reach), adding points to a reach or storage area, deleting points in a reach or storage area, deleting entire reaches, deleting junctions, numerically editing the reach schematic lines, and numerically editing the cross section schematic lines. Editing functions for the schematic

are found under the **Edit** menu of the geometric data window. When a specific editing function is selected, the interaction of the user with the schematic is restricted to performing that type of operation. When the user is finished performing that editing function they should turn off that editing function by selecting it again from the **Edit** menu. When none of the editing functions are turned on, the schematic goes back to the default mode of interaction. The default interaction mode for the schematic is described in the "Interacting with the Schematic" section of this document. A description of each editing function follows:

Change Name: This option allows the user to change the identifiers of any reach or junction. To change an identifier, you must be in the Change Name edit mode. This is accomplished by selecting the **Change Name** option from the **Edit** menu. Once you are in the Change Name edit mode, you then select the particular label that you would like to change by clicking the left mouse button over that label. When a label is selected, a pop up window will appear allowing you to enter a new label. The user can continue to change names by simply selecting the next label to be changed. The **Change Name** option can only be turned off by re-selecting it from the edit menu or by selecting any other edit option.

Move Object: This option allows you to move any label, junction, or point in a reach. This is accomplished by first selecting **Move Object** from the **Edit** menu, then selecting the particular object that you would like to move. To select an object and then move it, simply place the mouse pointer over the object, then press the left mouse button down. Move the object to the desired location and then release the left mouse button. The **Move Object** option will remain in effect until the user either turns it off (which is accomplished by re-selecting it) or selects any other edit option.

Add Points to a Reach od SA: This option allows the user to add additional points to the line that defines a reach or a storage area. This allows the user to make the schematic look more like the actual river system. To add additional points, first select **Add Points to a Reach or SA** from the **Edit** menu. Move the mouse pointer to the location in which you would like to add an additional point on the reach line, then click the left mouse button. After you have finished adding points to a reach, you can move them around by selecting the **Move Object** option from the **Edit** menu. To turn the "Add Points to a Reach or SA" mode off, simply re-select it from the Edit menu, or select any other edit function.

Remove Points in a Reach or SA: This option allows the user to remove points from a reach line or a storage area. To use this option, first select **Remove Points in a Reach or SA** from the **Edit** menu. Move the mouse pointer over the point that you would like to delete and then click the left mouse button. This option can only be turned off by either re-selecting the option from the Edit menu or by selecting another edit function.

Delete Reach: This option is used to delete a reach. This is accomplished by selecting the **Delete Reach** option from the **Edit** menu. A list box containing

all the available reaches will appear allowing you to select those reaches that you would like to delete. **Warning - Be careful when you delete reaches. When you delete a reach, all of its associated data will be deleted also.**

Delete Junction: This option is used to delete a junction. This is accomplished by selecting the **Delete Junction** option from the **Edit** menu. A list box containing all the available junctions will appear allowing you to select those junctions that you would like to delete.

Delete Storage Area: This option is used to delete a storage area. This is accomplished by selecting **Delete Storage Area** from the **Edit** menu. A selection box will appear allowing you to pick the storage areas that you would like to delete.

Delete Storage Connection: This option is used to delete a storage area connection. This is accomplished by selecting the **Delete Storage Conn** option from the **Edit** menu. A list box containing all the available storage area connections will appear allowing you to select the ones that you would like to delete.

Delete Pump Station: This option allows the user to select one or more pump stations to be deleted from the schematic. This is accomplished by selecting **Delete Pump Station** from the **Edit** menu. A list box containing all the available pump stations will appear allowing you to select the ones that you would like to delete.

Delete Nodes (XS, Bridges, Culverts, ...): This option allows the user to delete multiple locations at one time. For example, you can delete multiple cross sections at one time with this option. When this option is selected, a window will appear allowing you to select all of the nodes (cross sections, bridges, culverts, etc.) that you would like to delete.

Reach Schematic Lines: This option allows the user to numerically edit the coordinates of the river reach schematic lines. When the river system schematic is hand drawn on the screen, the coordinates of the river reach lines are put into a simple coordinate system that ranges from 0.0 to 1.0 in both the X and Y direction. However, the user has the option of taking real world coordinates (such as UTM or State Plane coordinates) off of a map and entering them into this table. If the user decides to use real world coordinates, real world coordinates must be added for all of the reaches of the schematic. If this is not done, the schematic will still be displayed in the simple 0.0 to 1.0 coordinate system (the hand drawn coordinates). Once real world coordinates have been entered for all of the river reaches, then the schematic will be drawn in that coordinate system. To enter/edit the reach schematic lines, select the **Reach Schematic Lines** option from the **Edit** menu. Once this option is selected, a window will appear allowing the user to enter/edit the coordinates of any of the reaches defined in the schematic.

XS Schematic Lines: This option allows the user to numerically edit the coordinates of the cross section schematic lines. When the river system is

hand drawn on the screen, the default coordinate system is a simple 0.0 to 1.0 range for both the X and Y direction. As cross sections are entered, they are automatically scaled based on the coordinates of the river reach line and the main channel distance between cross sections. Each cross section is drawn as a straight line perpendicular to the river reach schematic line. The user has the option of entering the real world coordinates (UTM or State Plane) of the cross section schematic lines. Each cross section schematic line must have at least two points, a start and an end, but additional points can be added if the cross section was taken as a multi segmented line. In order for the cross section schematic lines to be plotted in the real world coordinate system, the user must enter real world coordinates for all of the cross sections in the reach. To enter/edit the cross section schematic lines, select the **XS Schematic Lines** option from the **Edit** menu. Once this option is selected, a window will appear allowing the user to enter/edit the coordinates of any of the cross section schematic lines.

Storage Area Outlines: This option allows the user to enter/edit the X and Y coordinates that define the outline of a storage area. If you are using GIS data, these coordinates may be in the UTM or State Plane coordinate system. If you are not using GIS data, the coordinate system for the storage areas will just be in a 0 to 1.0 system. The coordinates of a storage area can also be edited graphically by using the **Move Object** option from the edit menu.

GIS Levee: This option allows the user to edit/enter the coordinates for a user specified levee. Levees can be defined in the GIS system and imported into HEC-RAS. The coordinates will consist of X, Y, and Z (elevation).

GIS Ineffective Flow Area: This option allows the user to edit/enter the coordinates for a user specified ineffective flow areas. Ineffective flow areas can be defined in the GIS system and imported into HEC-RAS. The coordinates will consist of X, Y, and Z (trigger elevation for when it will be turned off).

Hydraulic Connection Lines: This option allows the user to edit/enter coordinates for the line used to show a connection between two storage areas, between a storage area and a river reach, and between two river reaches. These lines cal also be edited graphically by using the **Move Object** option from the Edit menu.

Interacting With The Schematic

In addition to modifying the river schematic, there are options available to zoom in, zoom out, display the cross section river stationing, and reset the viewing extent of river system schematic. Additionally, the user has the ability to use the mouse to interact with the schematic. This is accomplished by moving the mouse pointer over an object (river reach line, junction, bridge, culvert, etc.) on the schematic and pressing down the left mouse button. Once the left mouse button is pressed down, a pop up menu will appear with options that are specific to that type of object. For example,

when the left mouse button is pressed down over a cross section, a menu will appear allowing the user to select options to: edit the cross section, plot the cross section, plot the profile for the reach that the cross section is in, display tabular output for the cross section, and plot the computed rating curve for that cross section. Another way of interacting with the schematic is to press the right mouse button while the mouse pointer is located any where over the schematic drawing area. This will bring up a pop up menu that is exactly the same as the View menu at the top of the drawing. This option is providing for convenience in getting to the View menu options. The options available from the **View** menu are as follows:

Zoom In: This option allows the user to zoom in on a piece of the schematic. This is accomplished by selecting **Zoom In** from the **View** menu, then specifying the area to zoom in on with the mouse. Defining the zoom area is accomplished by placing the mouse pointer in the upper left corner of the desired area. Then press down on the left mouse button and drag the mouse to define a box containing the desired zoom area. Finally, release the left mouse button and the viewing area will display the zoomed in schematic. Also displayed will be a small box in the upper right corner of the viewing area. This box will contain a picture of the entire schematic, with a rectangle defining the area that is zoomed in. In addition to showing you where you are at on the schematic, this zoom box allows you to move around the schematic without zooming out and then back in. To move the zoomed viewing area, simply hold down the left mouse button over the rectangle in the zoom box and move it around the schematic. The zoom box can also be resized. Resizing the zoom box is just like resizing a window.

Zoom Out: This option zooms out to an area that is twice the size of the currently zoomed in window. Zooming out is accomplished by selecting **Zoom Out** from the **View** menu on the geometric data window.

Full Plot: This option re-draws the plot to its full original size. The Full Plot option is accomplished by selecting **Full Plot** from the **View** menu on the geometric data window.

Pan: This option allows the user to move around when in a zoomed in mode. The pan option is accomplished by selecting **Pan** from the **View** menu of the geometric data window. When this option is selected, the mouse pointer will turn into a hand. Press the left mouse button and hold it down, then move the mouse. This will allow the user to move the zoomed in graphic. To turn the pan mode off, re-select the pan option from the view menu.

Set Schematic Plot Extents: This option allows the user to set the extents of the viewing area for the river system schematic. The user can enter a specific coordinate system, or utilize the default data system. The default data plot extents are from 0 to 1 for both the X and Y axis.

Find: This option allows the user to have the interface locate a specific feature on the schematic. This is especially useful when very large and complex river systems are being modeled.

Display River Stationing: This option allows you to display river station identifiers on the schematic. This is accomplished by selecting **Display River Stationing** from the **View** menu on the geometric data window.

Display Bank Stations: This option allows the user to display the main channel bank stations on the cross section lines of the schematic. This is accomplished by selecting **Display Bank Stations** from the **View** menu of the geometric data window.

Display Ineffective Areas: This option allows the user to display the location of ineffective flow areas on top of the cross section lines of the schematic. This is accomplished by selecting **Display Ineffective Areas** from the **View** menu of the geometric data window.

Display Levees: This option allows the user to display the location of levees on the cross section lines of the schematic. This is accomplished by selecting **Display Levees** from the **View** menu of the geometric data window.

Display XS Direction Arrows: This option allows the user to display arrows along the cross sections in the direction in which they were extracted. This option is useful when you have coordinates defined for the cross section, such that the software can detect the direction that the cross section was extracted. Cross-sections are suppose to be entered from left to right while looking downstream. If a cross section has not been entered in this manner, it should be reversed. HEC-RAS has an option to reverse the cross section stationing. This option can be found under the **Tools** menu bar of the geometric data editor. To display the cross section direction arrows, select **Display XS Direction Arrows** from the **View** menu of the geometric data window.

Fill in Storage Areas: This option allows the user to turn on and off the fill in color for the storage areas. Turning this off is very useful when a background picture is loaded.

Background Pictures.

Another option available to users is the ability to add background images for displayed behind the river system schematic. One or more pictures can be added. To enter a background picture, press the background picture button on the Tools menu bar (far right button). Select the add button to find a picture file to use as a background image. The software will automatically look for a companion file to the image, called a world file. This file contains information about the image, including the coordinate system and the extents of the image. The user has the option of rectifying the picture by entering coordinates for the left, right, top, and bottom sides of the picture, with respect to the coordinates of the river system schematic. If coordinates are not entered for the extent of the picture, the size of the picture will be based on its resolution and the resolution of your screen. Several graphical formats are supported for background maps, including: ESRI shape files (.shp); Cad files (.dgn, .dxf, .dwg); Windows bitmap (.bmp, .dip); Tiff and GeoTiff (.tif, .tff, .tiff); JFIF (.jpg, .jpeg); MrSID (.sid), and others.

Cross Section Data

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

Entering Cross Section Data

To enter cross section data, the user presses the **Cross Section** button on the Geometric Data window (Figure 6.1). Once the cross section button is pressed, the Cross Section Data Editor will appear as shown in Figure 6.2 (except yours will be blank until you have added some data). To add a cross section to the model, the user must do the following:

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

2. Go to the **Options** menu and select **Add a new Cross Section** from the list. An input box will appear prompting you to enter a river station identifier for the new cross section.

3. Enter all of the required data for the new cross section. Required data is the data that is openly displayed in the cross section editor window.

4. Enter any desired optional information (i.e., ineffective flow areas, levees, blocked obstructions, etc.). Optional cross section information is found under the **Options** menu.

5. Press the **Apply Data** button in order for the interface to accept the data. The apply data button does not save the data to the hard disk, it is used as a mechanism for telling the interface to use the information that was just entered. If you want the data to be saved to the hard disk you must do that from the **File** menu on the geometric data window.

The required information for a cross section consists of: the river, reach and river station identifiers; a description; X & Y coordinates (station and elevation points); downstream reach lengths; Manning's roughness coefficients; main channel bank stations; and contraction and expansion coefficients. All of the required information is displayed openly on the Cross Section Data editor (Figure 6.2). A description of this information follows:

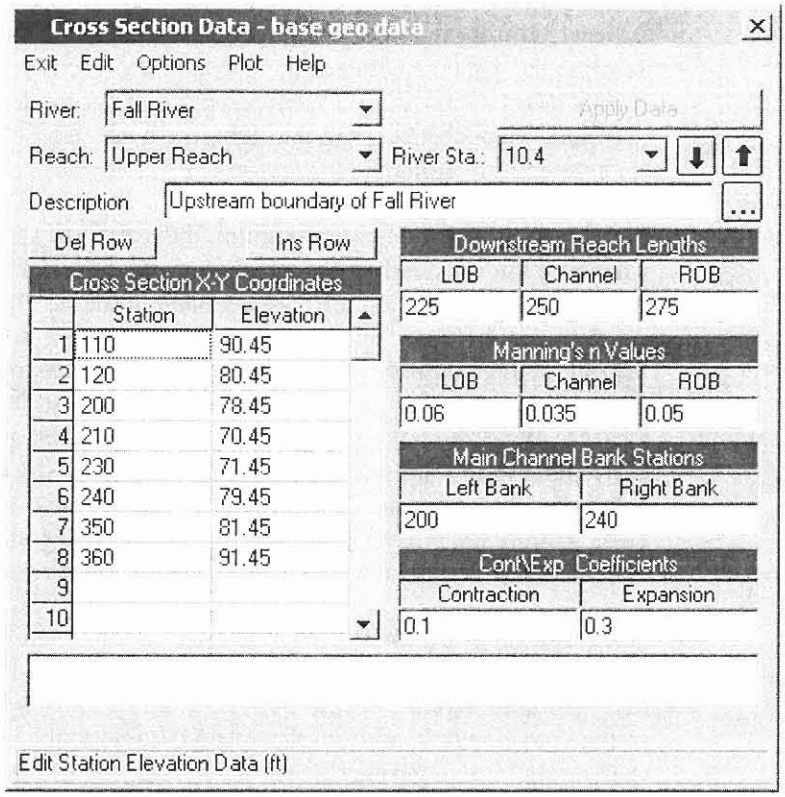


Figure 6.2 Cross Section Data Editor

River, Reach, and River Station. The River and Reach boxes allow the user to select a specific hydraulic reach from the available reaches in the schematic diagram. The river and reach labels define which reach the cross section will be located in. The River Station tag defines where the cross section will be located within the specified reach. The river station tag does not have to be the actual river station of the cross section, but it must be a numeric value. Cross sections are ordered in the reach from highest river station upstream to lowest river station downstream. The up and down arrow buttons next to the river station box can be used to sequentially move through the river stations.

Description. The description box is used to describe the cross section location in more detail than just the river, reach, and river station. This box has a limit of 512 characters. The first line of information is displayed, unless the button to the right of the box is pressed. Also, the first 40 characters of the description are used as a label for cross section plots and tables.

Cross Section X & Y Coordinates. This table is used to enter the station and elevation information of the cross section. Station and elevation information is entered in feet (meters for metric). **The cross section stationing (x-coordinates) are entered from left to right looking in the downstream direction**. Cross section stationing must be in increasing order. However, two or more stations can have the same value to represent vertical walls.

Downstream Reach Lengths. The downstream cross section reach lengths describe the distance between the current cross section and the next cross section downstream. Cross section reach lengths are defined for the left overbank, main channel, and the right overbank. Cross section reach lengths are entered in feet (meters for metric).

Manning's n Values. At a minimum, the user must specify Manning's n values for the left overbank, main channel, and the right overbank. Alternative roughness options are available from the **Options** menu.

Main Channel Bank Stations. The main channel bank stations are used to define what portion of the cross section is considered the main channel and what is considered left and right overbank area. The bank stations must correspond to stations entered on the cross section X & Y coordinates table. If the user enters a value that does not correspond to the station points of the cross section, the interface will ask the user if they would like the value to be automatically interpolated and added to the cross section data.

Contraction & Expansion Coefficients. Contraction and expansion coefficients are use to evaluate the amount of energy loss that occurs because of a flow contraction or expansion. The coefficients are multiplied by the change in velocity head from the current cross section to the next downstream cross section. In other words, the values entered at a particular cross section are used to compute losses that occur between that cross section and the next downstream cross section.

Once all of the required data for the cross section are entered, make sure you press the **Apply data** button to ensure that the interface accepts the data that was just entered.

Editing Cross Section Data

The bulk of the cross section data is the station and elevation information. There are several features available under the **Edit** menu to assist the user in modifying this information. These features include the following:

Undo Editing. This editing feature applies to all of the information on the cross section data editor. Once data has been entered and the **Apply Data** button has been pressed, the **Undo Editing** feature is activated. If any changes are made from this point, the user can get the original information back by selecting the **Undo Edit** option from the **Edit** menu. Once the **Apply Data** button is pressed, the new information is considered good and the **Undo Edit** feature is reset to the new data.

Cut, Copy, and Paste. Cut, Copy, and Paste features are available for the station and elevation information on the cross section editor. These features allow the user to pass cross section station and elevation data to and from the Windows Clipboard. To use this feature, first highlight a cell or multiple cells on the station and elevation table. Cells are highlighted by pressing down on the left mouse button and moving it over the cells that you would

like to be highlighted. Next select either the **Cut** or **Copy** feature from the **Edit** menu. If **Cut** is selected, the information is placed in the Windows Clipboard and then it is deleted from the table. If **Copy** is selected, the information is placed in the Windows Clipboard, but it also remains in the table. Once the information is in the Windows Clipboard it can be pasted into the station and elevation table of any cross section. To paste data into another cross section, first go to the cross section in which you would like the data to be placed. Highlight the area of the table in which you want the data to be placed. Then select the **Paste** option from the **Edit** menu. The cut, copy, and paste features can also be used to pass station and elevation information between HEC-RAS and other programs.

Delete. This option allows the user to delete a single cell or multiple cells in the station/elevation table. Once the cells are deleted, everything below those cells is automatically moved up. To use this option, first highlight the cells that you would like to delete, then select the **Delete** option from the **Edit** menu. If you would like to clear cells, without moving the data below those cells, simply highlight the cells and press the delete key.

Insert. This option allows the user to insert one or several rows in the middle of existing data in the station/elevation table. To use this option, first highlight the area in the table that you would like to be inserted. Then select **Insert** from the **Edit** menu. The rows will be inserted and all of the data will be moved down the appropriate number of rows. The user can also insert a single row by placing the curser in the row just below where you would like the new row to be inserted. Then select **Insert** from the **Edit** menu. The row will be inserted and all of the data below the current row will be moved down one row.

Cross Section Options

Information that is not required, but is optional, is available from the **Options** menu at the top of the cross section data editor window (Figure 6.2). Options consist of the following:

Add a new Cross Section. This option initiates the process of adding a cross section to the data set. The user is prompted to enter a river station tag for the new cross section. The river station tag locates the cross section within the selected reach. Once the river station is entered, the cross section data editor is cleared (except for some default values that get set) and the user can begin entering the data for the cross section. Whenever a new cross section is added to the data set, default values will appear for the contraction and expansion coefficients (0.1 and 0.3 respectively). Also, if the new cross section is not the first or most upstream cross section of the reach, the program will set default Manning's n values equal to the n values of the cross section just upstream of the new cross section. If the user does not want these default values, they can simply change them to whatever values they would like.

Copy Current Cross Section. This option allows the user to make a copy of the cross section that is currently displayed in the editor. When this option is selected, the user is prompted to select a river and reach for the new section, and then enter the a river station. Once the information is entered, the new cross section is displayed in the editor. At this point it is up to the user to change the description and any other information about the cross section. This option is normally used to make interpolated cross sections between two surveyed cross sections. Once the section is copied, the user can adjust the elevations and stationing of the cross section to adequately depict the geometry between the two surveyed sections.

Rename River Station. This option allows the user to change the River Station of the currently displayed cross section.

Delete Cross Section. This option will delete the currently displayed cross section. The user is prompted with a message stating specifically which cross section is going to be deleted, and requesting the user to press the **OK** button or the **Cancel** button. Once the OK button is pressed, the user will be prompted with a question of whether or not they would like the cross section reach lengths to be automatically adjusted to account for the removal of the cross section. If the user answers **YES** then the reach lengths of the current cross section, that is being deleted, will be added to the reach lengths of the next upstream cross section. If the user answers **NO**, then the cross section will be deleted with out adjusting any reach lengths.

Adjust Elevations. This option allows the user to adjust all of the elevations of the currently displayed cross section. Positive or negative elevation changes can be entered. Once the value is entered, the interface automatically adjusts all the elevations in the table.

Adjust Stations. This option allows the user to adjust the stationing of the currently displayed cross section. Two options are available. The first option **(Multiply by a Factor)** allows the user to separately expand and/or contract the left overbank, main channel, and the right overbank. When this option is selected, the user is prompted to enter a multiplier for each of the three flow elements (left overbank, main channel, and right overbank). If the multiplier is less than one, the flow element is contracted. If the multiplier is greater than one, the flow element is expanded. Once the information is entered, and the user hits the **OK** button, the interface automatically performs the contraction and/or expansions. The cross section should be reviewed to ensure that the desired adjustments were performed. The second option (**Add a Constant**) allows the user to add or subtract a constant value from all the stations in the cross section. This would allow the entire cross section to be shifted to the right or the left.

Adjust n or k Values. This option allows the user to either increase or decrease all the n or k values of the current cross section. The user is prompted for a single value. This value is then used as the multiplier for all of the n or k values of the current cross section.

Skew Cross Section. This option allows the user to adjust the stationing of a cross section based on a user entered skew angle. Cross-sections are suppose to be taken perpendicular to the flow lines. This may not always be the case, such as at bridges. In order for the program to use the correct flow area, the cross section stationing must be adjusted by taking the cosine of the skew angle times the stationing. When this option is selected, a window will appear allowing the user to enter a skew angle. Once the angle is entered, the software will automatically adjust the cross section stationing. The user can get back to the original stationing by putting a zero skew into the field.

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 or equal 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 Figure 6.3. The cross-hatched area on the left of the plot represents the ineffective flow area.

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 established 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. In other words, the program now assumes that the area will be conveying water in the downstream direction, such that it now uses that area in the conveyance calculations of the active flow area. However, the user has the option to set the ineffective flow areas to permanent, which will prevent them from turning off. When this option is used, water is allowed to go over top of the ineffective flow area.

The second option allows for the establishment of **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. Up to ten blocked ineffective flow areas can be entered at each cross section. Once the water surface goes above the elevation of the blocked ineffective flow area, the blocked area is no longer considered ineffective. However, the user has the option to set the blocked ineffective flow areas to permanent, which will prevent them from turning off. When this option is used, water is allowed to go over top of the ineffective flow area.

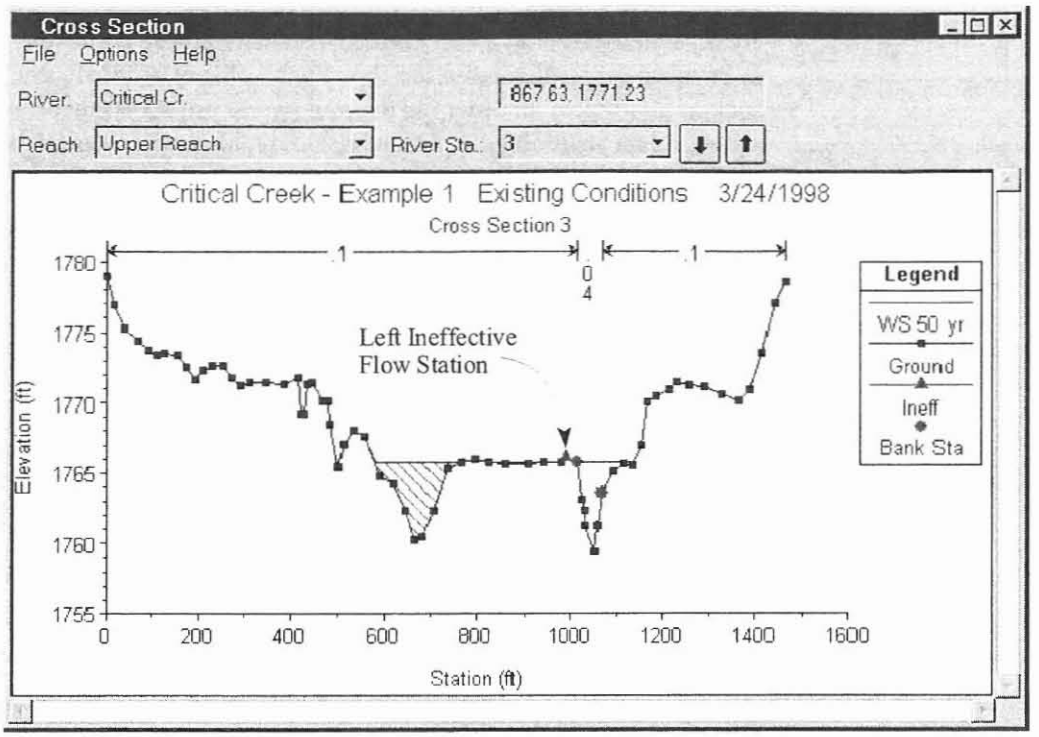


Figure 6.3 Cross section with ineffective flow areas

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. Levee stations must be defined explicitly, or the program assumes that water can go anywhere within the cross section. An example of a cross section with a levee on the left side is shown in Figure 6.4. In this example the levee station and elevation is associated with an existing point on the cross section.

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 when water comes into contact with the levee wall. An example of this is shown in Figure 6.5.

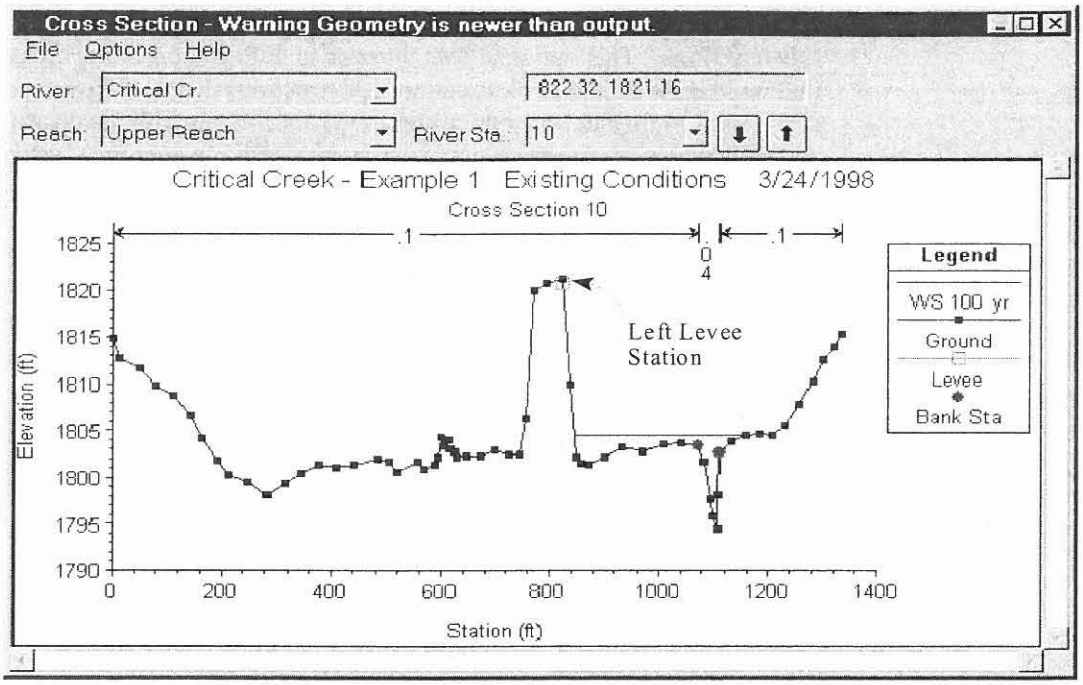


Figure 6.4 Example of the Levee Option

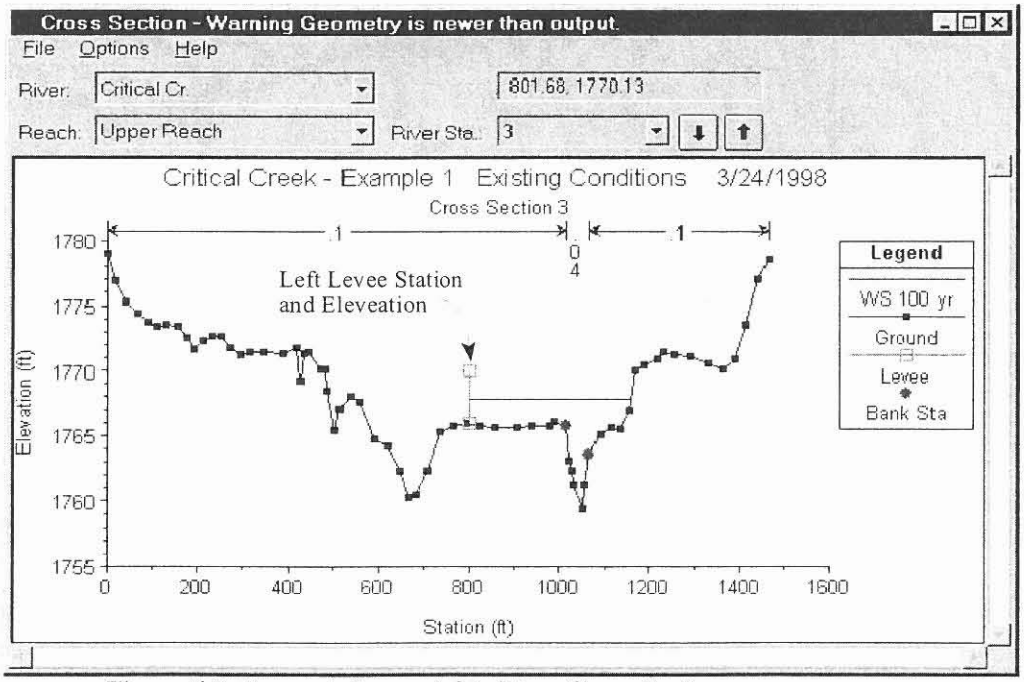


Figure 6.5 Example Levee Added to a Cross Section

Obstructions. This option allows the user to define areas of the cross section that will be permanently blocked out. Obstructions decrease flow area and add wetted perimeter when the water comes in contact with the obstruction. An obstruction does not prevent water from going outside of the obstruction.

Two alternatives are available for entering obstructions. The first option allows the user to define a left station and elevation and a right station and elevation (**normal obstructions**). 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. An example of this type of obstruction is shown in Figure 6.6.

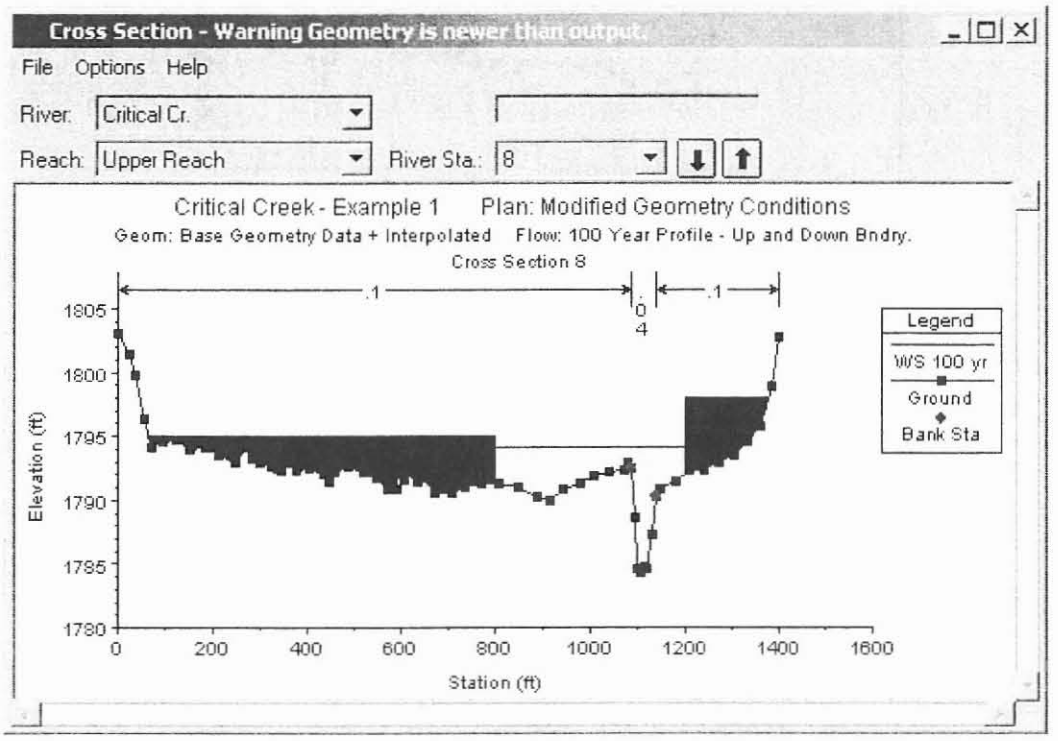


Figure 6.6 Example of Normal Obstructions

The second option, for obstructions, allows the user to enter up to 20 individual blocks (**blocked obstructions**). With this option the user enters a left station, a right station, and an elevation for each of the blocks. An example of a cross section with blocked obstructions is shown in Figure 6.7.

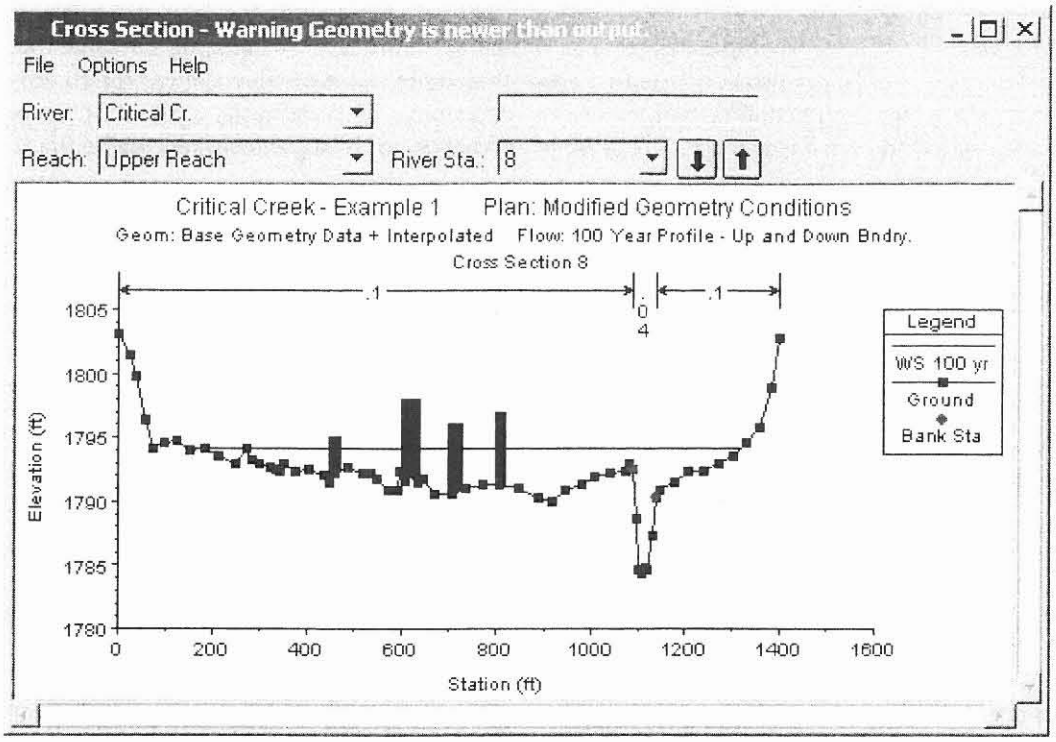


Figure 6.7 Example of a Cross Section With Blocked Obstruction

Add a Lid to XS. This option allows the user to add a lid (similar to a bridge deck/roadway) to any cross section. This is commonly used when trying to model a long tunnel. The ground geometry can be used to describe the bottom half of the tunnel, while the lid can describe the top half. A lid can be added to any number of cross sections in a row. The program treats cross sections with lids just like any other cross section. The energy equation is used to balance a water surface, with the assumption of open channel flow. The only difference is that the program will subtract out area and add wetted perimeter when the water surface comes into contact with the lid.

Add Ice Cover. This option allows the user to enter ice cover for the currently opened cross section. For a detailed discussion of ice cover, and ice modeling, please review the section called **Modeling Ice Cover** later in this chapter.

Add a Rating Curve. This option allows the user to add a rating curve to a cross section as an alternative to the program computing the water surface. The user is required to enter flow versus elevation information for the rating curve. When the program is executed in a steady flow mode, the program will interpolate a water surface elevation from the rating curve for the given flow of a particular profile.

Horizontal Variation in n Values. This option allows the user to enter more than three Manning's n values for the current cross section. When this option is selected, an additional column for n values is added to the cross section

coordinates table as shown in Figure 6.8. A Manning's n value must be placed in the first row of the table. This n value is good for all cross section stations until a new n value shows up in the table. The user does not have to enter an n value for every station, only at the locations where the n value is changing.

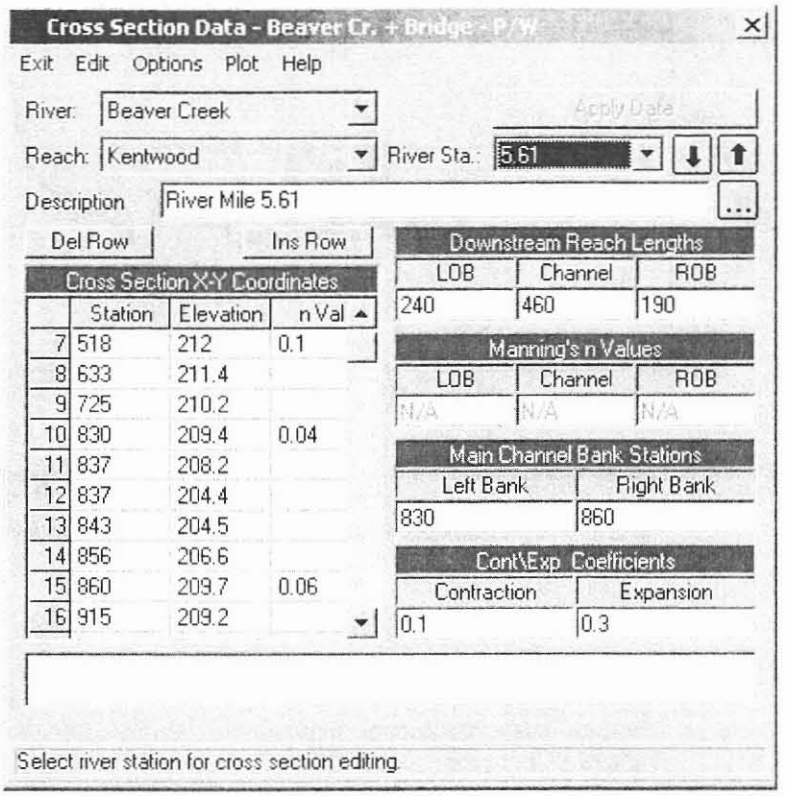
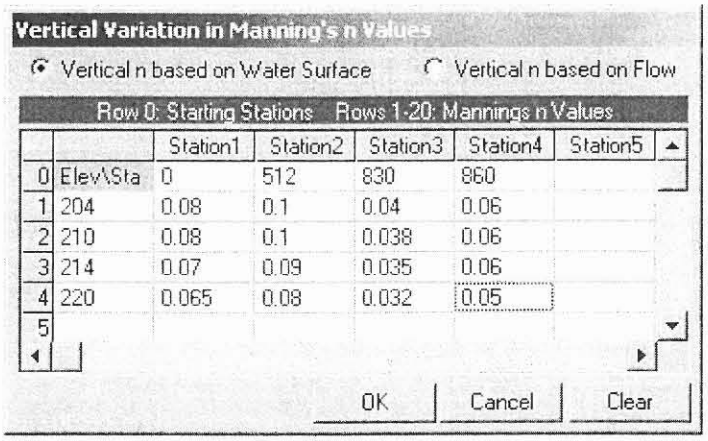


Figure 6.8 Cross section with horizontal variation of n values selected

Horizontal Variation in k Values. This option allows the user to enter k values (roughness heights) instead of n values. The k values are entered in the same manner as the horizontal variation of n values. To learn more about k values and how they are used in the program, see Chapter 3 of the Hydraulic Reference manual.

Vertical Variation in n Values. This option allows the user to enter Manning's n values that vary both horizontally as well as vertically. The user can vary the n value either by elevation or by flow. When this option is selected a window will appear as shown in Figure 6.9. The user enters the stationing for horizontal changes in n values across the top in row 0 (these stations are entered in the same manner as the horizontal variation of Manning's n value option). The elevations in which changes occur are entered in the first column. Then the actual Manning's n values are entered in rows 1-20 (columns 2-21). The program will interpolate Manning's n values whenever the actual water surface is between the entered elevations. If the

water surface is below the first elevation entered, then the values from that elevation will be used. Likewise, if the water surface is above the last elevation entered, the program will use the n values from the last elevation specified. No extrapolation is done on either side of the user entered values.



		Station1	Station2	Station3	Station4	Station5
0	Elev\Sta	0	512	830	860	
1	204	0.08	0.1	0.04	0.06	
2	210	0.08	0.1	0.038	0.06	
3	214	0.07	0.09	0.035	0.06	
4	220	0.065	0.08	0.032	0.05	
5						

Figure 6.9 Vertical Variation of Manning's n Values Window

Plotting Cross Section Data

Once all the data have been entered for a cross section, you should plot the cross section to inspect it for possible data errors. To plot the current cross section from the cross section editor, select **Plot Cross Section** from the **Plot** menu.

Stream Junctions

Entering Junction Data

Stream junctions are defined as locations where two or more streams come together or split apart. Junction data consist of a description, reach lengths across the junction, tributary angles, and modeling approach. To enter junction data the user presses the **Junction** button on the Geometric Data window (Figure 6.1). Once the junction button is pressed, the junction editor will appear as shown in Figure 6.10.

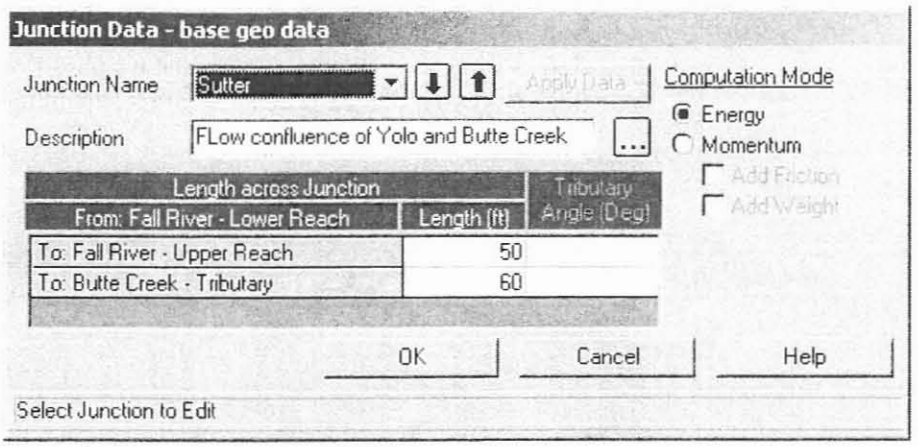


Figure 6.10 Junction Data Editor

The junction editor will come up with one of the junctions loaded. Fill out the description and reach lengths for the junction. Reach lengths across the junction are entered here instead of the cross section data editor. This allows for the lengths across very complicated confluences (i.e., flow splits) to be accommodated. In the cross section data, the reach lengths for the downstream cross section of each reach should be left blank or set to zero.

Selecting A Modeling Approach

In HEC-RAS a junction can be modeled by either the energy equation or the momentum equation. The energy equation does not take into account the angle of a tributary coming in or leaving, while the momentum equation does. In most cases the amount of energy loss due to the angle of the tributary flow is not significant, and using the energy equation to model the junction is more than adequate. However, 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 additional column is added to the table next to the junction lengths. This column is used to enter an angle for any river reach that is coming into or exiting the main river. For the reaches that are considered to be the main river, the angle should be left blank or set to zero. Also, the user has the option to turn friction and weight forces on or off during the momentum calculations. The default is to have the weight force turned off.

If there is more than one junction in the river schematic, the other junctions can be selected from the Junction Name box at the upper left corner of the window. Enter all the data for each junction in the river system, then close the window by pressing the **OK** button in the lower left corner of the window. When the junction data editor is closed the data are automatically applied.

Bridges and Culverts

Once all of the necessary cross-section data have been entered, the modeler can then add any bridges or culverts that are required. HEC-RAS computes energy losses caused by structures such as bridges and culverts in three parts. One part consists of losses that occur in the reach immediately downstream from the structure where an expansion of flow takes place. The second part is the losses at the structure itself, which can be modeled with several different methods. The third part consists of losses that occur in the reach immediately upstream of the structure where the flow is contracting to get through the opening.

The bridge routines in HEC-RAS allow the modeler to analyze a bridge with several different methods without changing the bridge geometry. The bridge routines have the ability to model low flow (Class A, B, and C), low flow and weir flow (with adjustments for submergence), pressure flow (orifice and sluice gate equations), pressure and weir flow, and high flows with the energy equation only. The model allows for multiple bridge and/or culvert openings at a single location.

The culvert hydraulics in HEC-RAS are based on the Federal Highway Administrations (FHWA) standard equations from the publication Hydraulic Design of Highway Culverts (FHWA, 1985). The culvert routines include the ability to model circular, box, elliptical, arch, pipe arch, low profile arch, high profile arch, and semi circular culverts. The HEC-RAS program has the ability to model multiple culverts at a single location. The culverts can have different shapes, sizes, elevations, and loss coefficients. The user can also specify the number of identical barrels for each culvert type.

Cross Section Locations

The bridge and culvert routines utilize four user defined cross sections in the computations of energy losses due to the structure. A plan view of the basic cross section layout is shown in Figure 6.11.

Cross section 1 is located sufficiently downstream from the structure so that the flow is not affected by the structure (i.e., the flow has fully expanded). This distance should generally be determined by field investigation during high flows. Generally, field investigation during high flows is not possible. The expansion distance will vary depending upon the degree of constriction, the shape of the constriction, the magnitude of the flow, and the velocity of the flow. Table 6.1 offers ranges of expansion ratios, which can be used for different degrees of constriction, different slopes, and different ratios of the overbank roughness to main channel roughness. Once an expansion ratio is selected, the distance to the downstream end of the expansion reach (the distance L_e on Figure 6.11) is found by multiplying the expansion ratio by the average obstruction length (the average of the distances A to B and C to D from Figure 6.11).

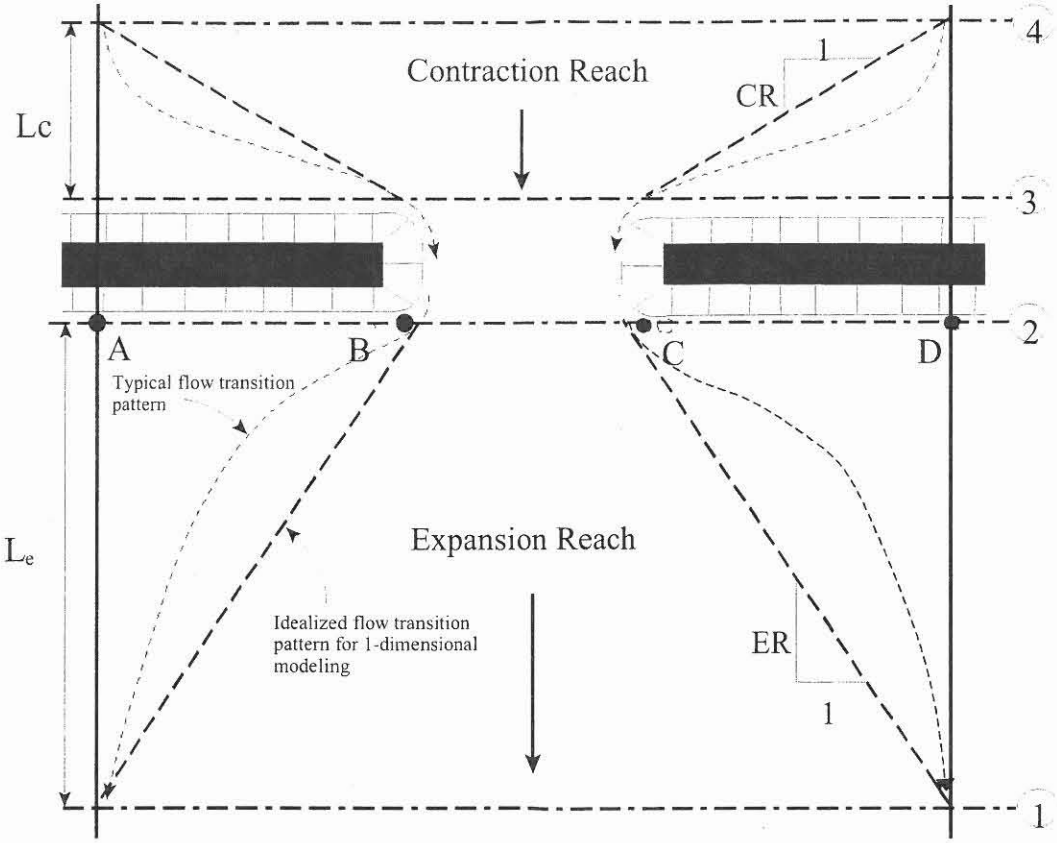


Figure 6.11 Cross Section Locations at a Bridge or Culvert

The average obstruction length is half of the total reduction in floodplain width caused by the two bridge approach embankments. In Table 6.1, b/B is the ratio of the bridge opening width to the total floodplain width, n_{ob} is the Manning n value for the overbank, n_c is the n value for the main channel, and S is the longitudinal slope. The values in the interior of the table are the ranges of the expansion ratio. For each range, the higher value is typically associated with a higher discharge.

Table 6.1
Ranges of Expansion Ratios

		$n_{ob} / n_{c=1}$	$n_{ob} / n_{c=2}$	$n_{ob} / n_{c=4}$
b/B = 0.10	S = 1 ft/mile	1.4 – 3.6	1.3 – 3.0	1.2 – 2.1
	5 ft/mile	1.0 – 2.5	0.8 – 2.0	0.8 – 2.0
	10 ft/mile	1.0 – 2.2	0.8 – 2.0	0.8 – 2.0
b/B = 0.25	S = 1 ft/mile	1.6 – 3.0	1.4 – 2.5	1.2 – 2.0
	5 ft/mile	1.5 – 2.5	1.3 – 2.0	1.3 – 2.0
	10 ft/mile	1.5 – 2.0	1.3 – 2.0	1.3 – 2.0
b/B = 0.50	S = 1 ft/mile	1.4 – 2.6	1.3 – 1.9	1.2 – 1.4
	5 ft/mile	1.3 – 2.1	1.2 – 1.6	1.0 – 1.4
	10 ft/mile	1.3 – 2.0	1.2 – 1.5	1.0 – 1.4

A detailed study of flow contraction and expansions at bridges was undertaken by the Hydrologic Engineering Center. The results of this study have been published as a research document entitled "Flow Transitions in Bridge Backwater Analysis" (RD-42 HEC, 1995). The purpose of this study was to provide better guidance to hydraulic engineers performing water surface profile computations through bridges. Specifically the study focused on determining the expansion reach length, L_e; the contraction reach length, L_c; the expansion energy loss coefficient, C_e; and the contraction energy loss coefficient, C_c. A summary of this research, and the final recommendations, can be found in Appendix B of the HEC-RAS Hydraulic Reference manual.

The user should not allow the distance between cross section 1 and 2 to become so great that friction losses will not be adequately modeled. If the modeler feels that the expansion reach will require a long distance, then intermediate cross sections should be placed within the expansion reach in order to adequately model friction losses. The user will need to estimate ineffective flow areas for these intermediate cross sections.

Cross section 2 is located immediately downstream from the bridge (i.e., within a short distance). This cross section should represent the natural ground just outside the bridge. This section is normally located at the toe of the downstream bridge embankment.

Cross section 3 should be located just upstream from the bridge. The distance between cross section 3 and the bridge should be relatively short. This distance should only reflect the length required for the abrupt acceleration and contraction of the flow that occurs in the immediate area of the opening. Cross section 3 represents the natural ground just upstream of the bridge. This section is normally located at the toe of the upstream bridge embankment.

Both cross sections 2 and 3 will have ineffective flow areas to either side of the bridge opening during low flow and pressure flow. In order to model only the effective flow areas at these two sections, the modeler should use the ineffective flow area option. This option is selected from the cross section data editor. For a detailed discussion of how to set the ineffective flow area stations and elevations, see Chapter 5 of the Hydraulic Reference manual.

Cross section 4 is an upstream cross section where the flow lines are approximately parallel and the cross section is fully effective. In general, flow contractions occur over a shorter distance than flow expansions. The distance between cross section 3 and 4 (the contraction reach length, L_c) should generally be determined by field investigation during high flows. Traditionally, the Corps of Engineers recommends locating the upstream cross section a distance equal to one times the average length of the side constriction caused by the structure abutments. The contraction distance will vary depending upon the degree of constriction, the shape of the constriction, the magnitude of the flow, and the velocity of the flow. As mentioned previously, the detailed study "Flow Transitions in Bridge Backwater Analysis" (RD-42, HEC, 1995) was performed to provide better guidance to

hydraulic engineers performing water surface profile computations through bridges. A summary of this research, and the final recommendations, can be found in Appendix B of the HEC-RAS Hydraulic Reference manual.

When the user adds a bridge at a particular river station, the program automatically formulates two additional cross sections inside of the bridge structure. The geometry inside of the bridge is a combination of the bounding cross sections (2 and 3) and the bridge geometry. The bridge geometry consists of the bridge deck, abutments if necessary, and any piers that may exist. The user can specify different bridge geometry for the upstream and downstream sides of the structure if necessary. Cross section 2 and the structure information on the downstream side are used as the geometry just inside the structure at the downstream end. Cross section 3 and the upstream structure information are used as the bridge geometry just inside the structure at the upstream end. The user has the option to edit these internal bridge cross sections, in order to make adjustments to the geometry.

For a more detailed discussion on laying out cross sections around bridges, the user is referred to chapter 5 of the Hydraulic Reference Manual.

Contraction and Expansion Losses

Losses due to the contraction and expansion of flow between cross sections are determined during the standard step profile calculations. Contraction and Expansion losses are described in terms of coefficient times the absolute value of the change in velocity head between adjacent cross sections. When the velocity head increases in the downstream direction a contraction coefficient is used; and when the velocity head decreases in the downstream direction, an expansion coefficient is used. For a detailed discussion on selecting contraction and expansion coefficients at bridges, the user is referred to chapter 5 of the HEC-RAS Hydraulic Reference Manual.

Bridge Hydraulic Computations

Low Flow Computations. For low flow computations the program first uses the momentum equation to identify the class of flow. This is accomplished by first calculating the momentum at critical depth inside the bridge at the upstream and downstream ends. The end with the higher momentum (therefore most constricted section) will be the controlling section in the bridge. The momentum at critical depth in the controlling section is then compared to the momentum of the flow downstream of the bridge when performing a subcritical profile (upstream of the bridge for a supercritical profile). If the momentum downstream is greater than the critical depth momentum inside the bridge, the class of flow is considered to be completely subcritical (i.e., class A low flow). If the momentum downstream is less than the momentum at critical depth in the bridge, then it is assumed that the constriction will cause the flow to pass through critical depth and a hydraulic jump will occur at some distance downstream (i.e., class B low flow). If the profile is completely supercritical through the bridge then this is class C low flow. Depending on the class of flow the program will do the following:

Class A low flow. Class A low flow exists when the water surface through the bridge is completely subcritical (i.e., above critical depth). Energy losses through the expansion (sections 2 to 1) are calculated as friction losses and expansion losses. Friction losses are based on a weighted friction slope times a weighted reach length between sections 1 and 2. The average friction slope is based on one of the four available alternatives in HEC-RAS, with the average-conveyance method being the default. This option is user selectable. The average length used in the calculation is based on a discharge-weighted reach length.

There are four methods for computing losses through the bridge (from 2 to 3):

- Energy equation (standard step method)
- Momentum balance
- Yarnell equation
- FHWA WSPRO method

The user can select any or all of these methods in the computations. If more than one method is selected, the user must choose either a single method as the final solution or tell the program to use the method that computes the greatest energy loss through the bridge as the answer at section 3. This allows the modeler to compare the answers from several techniques all in a single execution of the program. Minimal results are available for all the methods computed, but detailed results are available for the method that is selected as the final answer.

Energy losses through the contraction (sections 3 to 4) are calculated as friction losses and contraction losses. Friction and contraction losses between sections 3 and 4 are calculated the same as friction and expansion losses between sections 1 and 2.

Class B low flow. Class B low flow can exist for either subcritical or supercritical profiles. For either profile, class B flow occurs when the profile passes through critical depth in the bridge constriction. For a **subcritical profile**, the momentum equation is used to compute an upstream water surface above critical depth and a downstream water surface below critical depth, using a momentum balance through the bridge. For a **supercritical profile**, the bridge is acting as a control and is causing the upstream water surface elevation to be above critical depth. Momentum is used again to calculate an upstream water surface above critical depth and a downstream water surface below critical depth. The program will proceed with forewater calculations downstream from the bridge.

Class C low flow. Class C low flow exists when the water surface through the bridge is completely supercritical. The program can use either the energy or the momentum equation to compute the water surface through the bridge.

Pressure Flow Computations. Pressure flow occurs when the flow comes into contact with the low chord of the bridge. Once the flow comes into contact with the upstream side of the bridge, a backwater occurs and orifice flow is established. The program will handle two cases of orifice flow: the first is when only the upstream side of the bridge is in contact with the water; and the second is when the bridge constriction is flowing completely full. For the first case, a sluice gate type of equation is used, as described in "Hydraulics of Bridge Waterways" (FHWA, 1978). In the second case, the standard full flowing orifice equation is used. The program will begin checking for the possibility of pressure flow when the energy grade line goes above the maximum low chord elevation. Once pressure flow is computed, the pressure flow answer is compared to the low flow answer and the higher of the two is used. The user has the option to tell the program to use the water surface, instead of energy, to trigger the pressure flow calculation.

Weir Flow Computations. Flow over the bridge and the roadway approaching the bridge will be calculated using the standard weir equation. For high tailwater elevations the program will automatically reduce the amount of weir flow to account for submergence on the weir. This is accomplished by reducing the weir coefficient based on the amount of submergence. When the weir becomes highly submerged, the program will automatically switch to calculating losses based on the energy equation (standard step backwater). The criteria for when the program switches to energy based calculations is user controllable.

Combination Flow. Sometimes combinations of low flow or pressure flow occur with weir flow. In these cases an iterative procedure is used to determine the amount of each type of flow.

Entering and Editing Bridge Data

To enter bridge data the user presses the **Bridge/Culvert** button on the geometric data window (Figure 6.1). Once the bridge/culvert button is pressed, the Bridge/Culvert Data Editor will appear as shown in Figure 6.12 (your bridge/culvert editor will come up with a blank window until you have entered the bridge data). To add a bridge to the model, do the following:

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

2. Go to the **Options** menu and select **Add a Bridge and/or Culvert** from the list. An input box will appear prompting you to enter a river station identifier for the new bridge.

3. Enter all of the required data for the new bridge. This includes:
   - Bridge Deck
   - Sloping Abutments (optional)
   - Piers (optional)
   - Bridge modeling approach information

4. Enter any desired optional information. Optional bridge information is found under the **Options** menu at the top of the window.

5. Press the **Apply Data** button for the interface to accept the data.

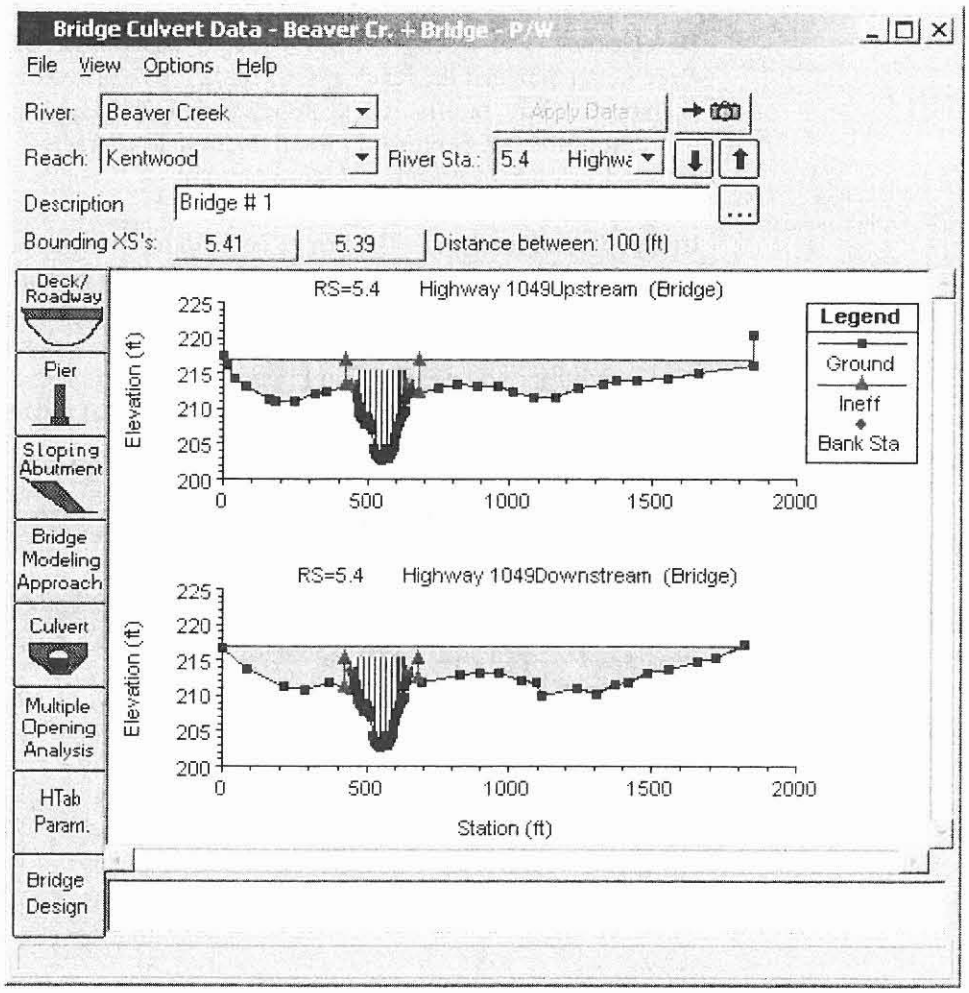


Figure 6.12 Bridge/Culvert Data Editor

The required information for a bridge consists of: the river, reach, and river station identifiers; a short description of the bridge; the bridge deck; bridge abutments (if they exist); bridge piers (if the bridge has piers); and specifying the bridge modeling approach. A description of this information follows:

River, Reach and River Station. The River and Reach boxes allow the user to select a river and reach from the available reaches that are defined in the schematic diagram. The reach label defines which reach the bridge will be located in. The River Station tag defines where the bridge will be located within the specified reach. The river station tag does not have to be the actual river station of the bridge, but it must be a numeric value. The river station tag for the bridge should be numerically between the two cross sections that

bound the bridge. Once the user selects **Add a Bridge and/or Culvert** from the options menu, an input box will appear prompting you to enter a river station tag for the new bridge. After the river station tag is entered, the two cross sections that bound the bridge will be displayed on the editor.

Description. The description box is used to describe the bridge location in more detail than just the reach and river station. This box has a limit of 256 characters. Only the first line of information is displayed, unless the button to the right of the box is pressed. Also, the first 40 characters of the description are used as a label for bridge plots and tables.

Bridge Deck/Roadway. The bridge deck editor is used to describe the area that will be blocked out due to the bridge deck, road embankment and vertical abutments. To enter bridge deck information the user presses the **Deck** button on the Bridge/Culvert Data Editor. Once the deck button is pressed, the Deck Editor will appear as in Figure 6.13 (except yours will be blank). The information entered in the deck editor consists of the following:

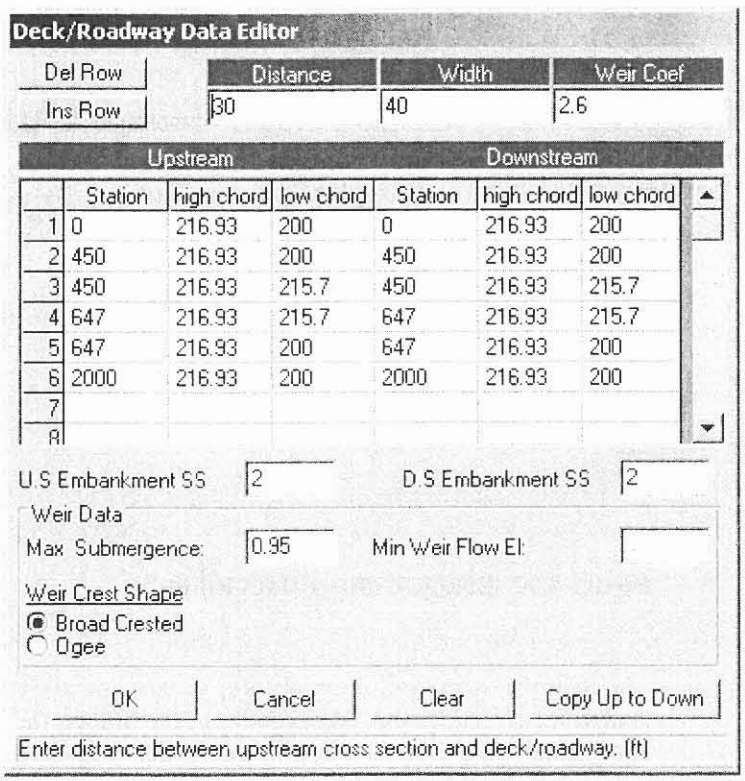


Figure 6.13 Bridge Deck/Roadway Data Editor

Distance - The distance field is used to enter the distance between the upstream side of the bridge deck and the cross section immediately upstream of the bridge (see Figure 6.14, "Upstream Distance"). This distance is entered in feet (or meters for metric).

Width - The width field is used to enter the width of the bridge deck along the stream (Figure 6.14, "Bridge Width"). The distance between the bridge deck and the downstream bounding cross section will equal the main channel reach length minus the sum of the bridge "width" and the "distance" between the bridge and the upstream section. The width of the bridge deck should be entered in feet (meters for metric).

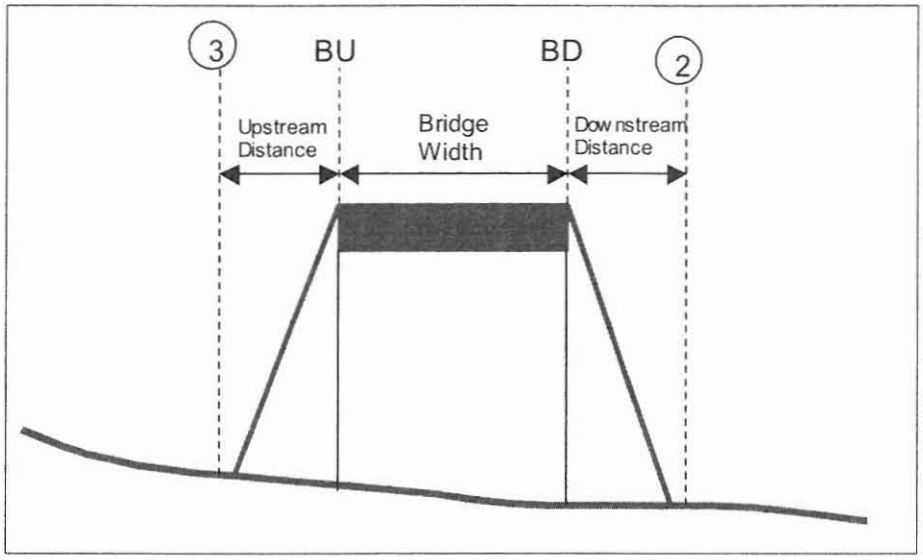


Figure 6.14 Bridge Profile With Upstream Distance, Bridge Width, and Downstream Distance.

Weir Coefficient - Coefficient that will be used for weir flow over the bridge deck in the standard weir equation.

Upstream Stationing, High Chord, and Low Chord - This table is used to define the geometry of the bridge deck on the upstream side of the bridge. The information is entered from left to right in cross section stationing. The deck is the area between the high and low chord elevation information. The stationing of the deck does not have to equal the stations in the bounding cross section, but it must be based on the same origin. The **Del Row** and **Ins Row** buttons allow the user to delete and insert rows.

Downstream Stationing, High Chord, and Low Chord - This portion of the table is used to define the geometry of the bridge deck on the downstream side of the bridge. If the geometry of the downstream side is the same as the upstream side, then the user only needs to press the **Copy Up to Down** button. When this button is pressed, all of the upstream bridge deck information is copied to the downstream side. If the bridge deck information on the downstream side is different than the upstream side, then the user must enter the information into the table.

U.S. Embankment SS - This field is used to enter the slope of the road embankment on the upstream side of the bridge. The slope should be entered as the horizontal to vertical distance ratio of the embankment. This variable

is generally not used in the computations, but is used for display purposes in the profile plot. However, if the user has selected the FHWA WSPRO bridge method for low flow, this field will be used in the computation of the bridge discharge coefficient.

D.S. Embankment SS - This field is used to enter the slope of the road embankment on the downstream side of the bridge. The slope should be entered as the horizontal to vertical distance ratio of the embankment. This variable is generally not used in the computations, but is used for display purposes in the profile plot. However, if the user has selected the FHWA WSPRO bridge method for low flow, this field will be used in the computation of the bridge discharge coefficient.

Max Allowable Submergence - The maximum allowable submergence ratio that can occur during weir flow calculations over the bridge deck. If this ratio is exceeded, the program automatically switches to energy based calculations rather than pressure and weir flow. The default value is 0.95 (95 percent submerged).

Submergence Criteria - When submergence occurs there are two choices available to figure out how much the weir coefficient should be reduced due to the submergence. The first method is based on work that was done on a trapezoidal shaped broad crested weir (FHWA, 1978). The second criterion was developed for an Ogee spillway shape (COE,1965). The user should pick the criterion that best matches their problem.

Min Weir Flow El - This field is used to set the minimum elevation for which weir flow will begin to be evaluated. Once the computed upstream energy becomes higher than this elevation, the program begins to calculate weir flow. However, the weir flow calculations are still based on the actual geometry of the deck/roadway, and are not effected by this elevation. If this field is left blank, the elevation that triggers weir flow is based on the lowest high chord elevation on the upstream side of the bridge deck. Also, weir flow is based on the elevation of the energy grade line and not the water surface.

Once all of the bridge deck information is entered, the user should press the **OK** button at the bottom of the window. Pressing the **OK** button tells the interface to accept the data and close the window. Once the deck editor closes, the graphic of the bridge deck will appear on the Bridge/Culvert Data window. An example of this is shown in Figure 6.15. **Note! The data are not saved to the hard disk at this point.** Geometric data can only be saved to the hard disk from the **File** menu of the Geometric Data window.

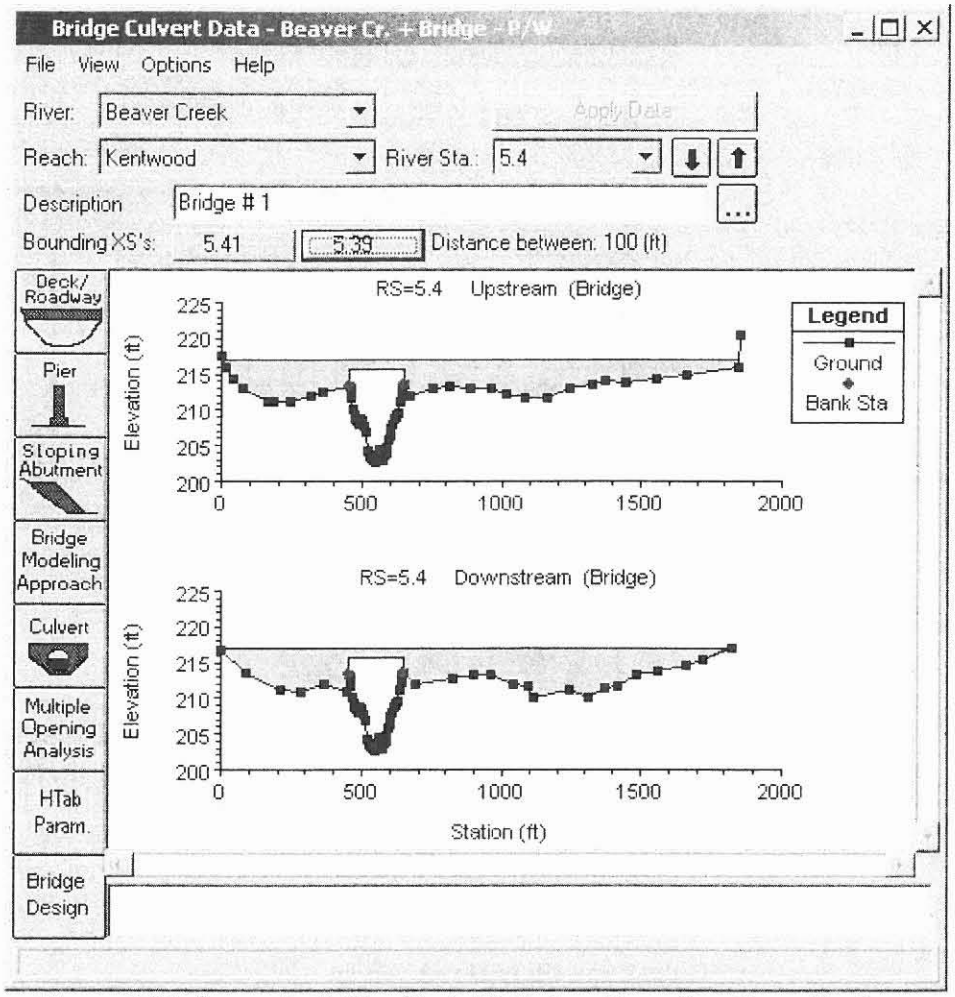


Figure 6.15 Example Bridge Deck Plotted on Bounding Cross Sections

Sloping Bridge Abutments. The sloping bridge abutments editor is used to supplement the bridge deck information. Whenever bridge abutments are protruding towards the main channel (sloping inward abutments), it will be necessary to block out additional area that cannot be accounted for in the bridge deck/roadway editor. If the bridge has vertical wall abutments, then it is not necessary to use this editor. Vertical wall abutments can be included as part of the bridge deck/roadway data. To add sloping abutments, the user presses the **Sloping Abutment** button on the Bridge/Culvert Data editor. Once this button is pressed the Abutment data editor will appear as in Figure 6.16.

Sloping abutments are entered in a similar manner to the bridge deck/roadway. When the editor is open, it has already established an abutment # of 1. Generally a left and right abutment are entered for each bridge opening. Sloping abutment data are entered from left to right, looking in the downstream direction. In general it is usually only necessary to enter two points to describe each abutment.

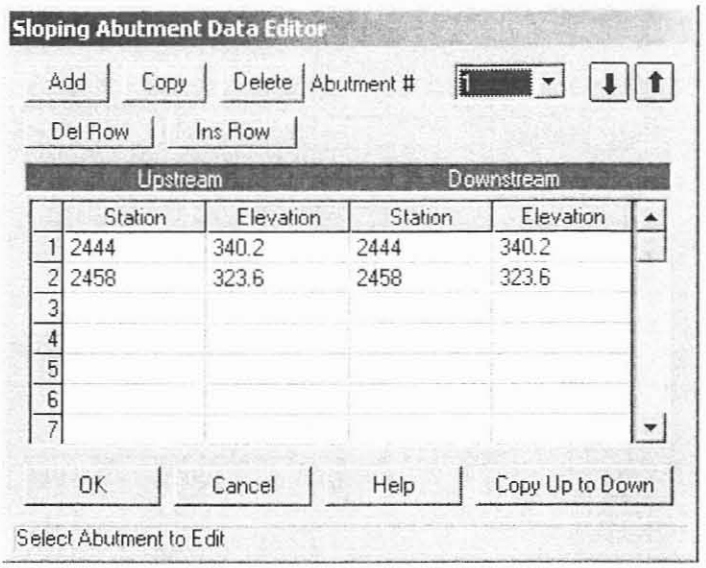


Figure 6.16 Abutment Data editor

The data for each abutment consist of a skew angle (this is optional) and the station and elevation information. The station and elevation information represents the high chord information of the abutment. The low chord information of the abutment is assumed to be below the ground, and it is therefore not necessary to enter it. The geometric information for each abutment can vary from upstream to downstream. If this information is the same, then the user only needs to enter the upstream geometry and then press the **Copy Up to Down** button.

To add additional sloping abutments, the user can either press the **ADD** or the **Copy** button. To delete an abutment, press the **Delete** button. Once all of the abutment data are entered, the user should press the **OK** button. When the **OK** button is pressed, the abutment information is accepted and the editor is closed. The abutments are then added to the bridge graphic on the Bridge/Culvert Data editor. An example of a sloping bridge abutment is shown in Figure 6.17. This graphic is zoomed in on the left abutment of the bridge.

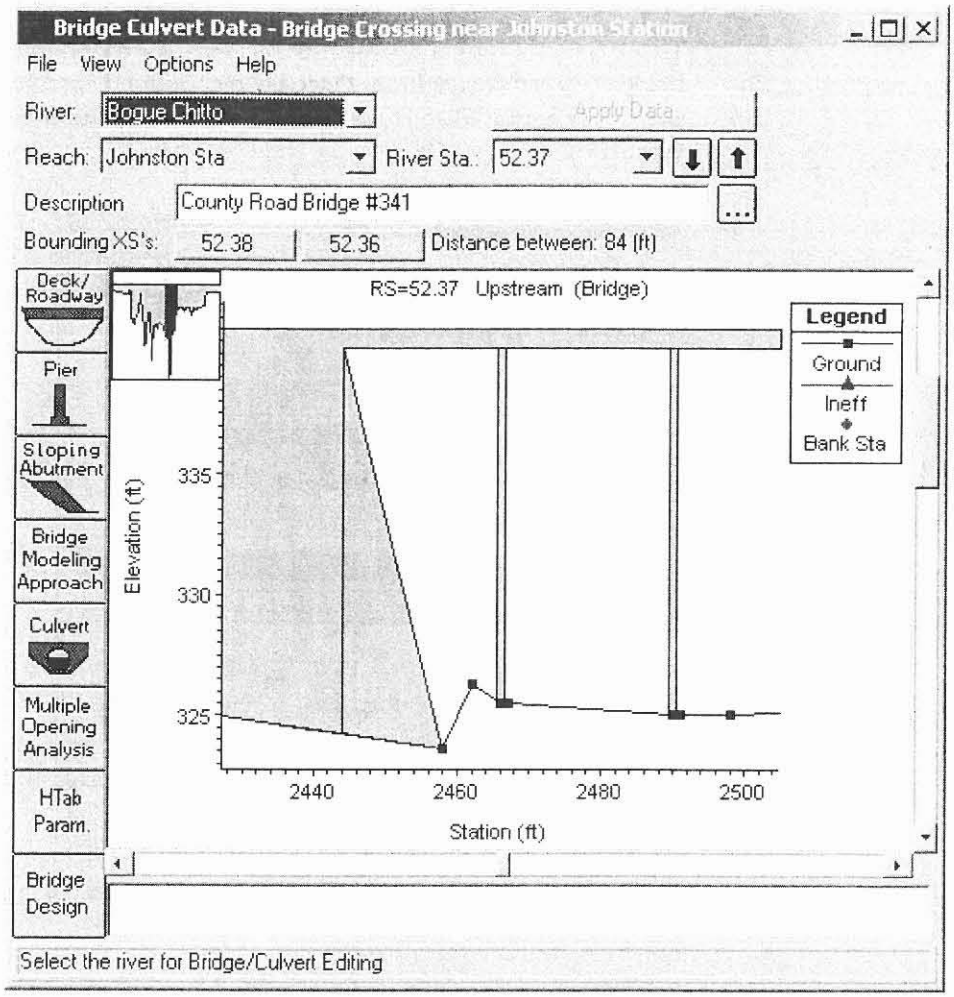


Figure 6.17 Example of a Sloping Abutment

Bridge Piers. The bridge pier editor is used to describe any piers that exist in the bridge opening. **Note! All piers must be entered through the Pier Editor, they should not be included as part of the ground or bridge deck.** Several of the low flow bridge computations require that the piers be defined separately in order to determine that amount of area under the water surface that is blocked by the piers. If the piers are included with the ground or the bridge deck, several of the methods will not compute the correct amount of energy loss for the piers.

To enter pier information, the user presses the **Pier** button on the Bridge/Culvert Data editor. Once the pier button is pressed, the pier data editor will appear as in Figure 6.18 (except yours will not have any data in it yet).

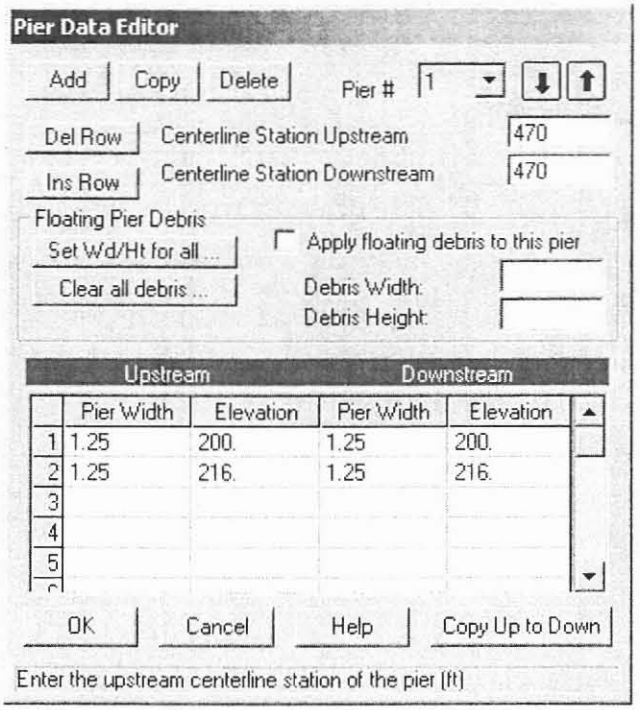


Figure 6.18 Pier Data Editor

When the pier data editor appears it will have already defined the first pier as pier # 1. The user is required to enter a centerline station for both the upstream and downstream side of the pier. The skew angle is entered in degrees that the pier is skewed from a line parallel to the flow. The skew angle is an optional item. The pier geometry is entered as pier widths and elevations. The elevations must start at the lowest value and go to the highest value. Generally the elevations should start below the ground level. Any pier area below the ground will be clipped off automatically. Pier widths that change at a single elevation are handled by entering two widths at the same elevation. The order of the widths in the table is very important. Keep in mind that the pier is defined from the ground up to the deck. If the pier geometry on the downstream side is the same as the upstream side, simply press the **Copy Up to Down** button after the upstream side data are entered.

The user also has the option of defining floating pier debris. If the **Floating Debris** option is selected, the user will need to enter a width and a height for the debris. The user can set a different height and width of debris for each pier, or there is a button that will allow the user to enter a single height and width that will be used for all of the piers.

Additional piers can be added by pressing either the **Add** or the **Copy** button. If the piers are the same shape, it is easier to use the copy button and simply change the centerline stations of the new pier. To delete a pier, simply press the **Delete** button and the currently displayed pier will be deleted. Once all of the pier data are entered, press the **OK** button. When the OK button is pressed, the data will be accepted and the pier editor will be closed. The graphic of the bridge will then be updated to include the piers. An example bridge with piers is shown in Figure 6.19. This graphic is only the upstream side of the bridge with a zoomed in view.

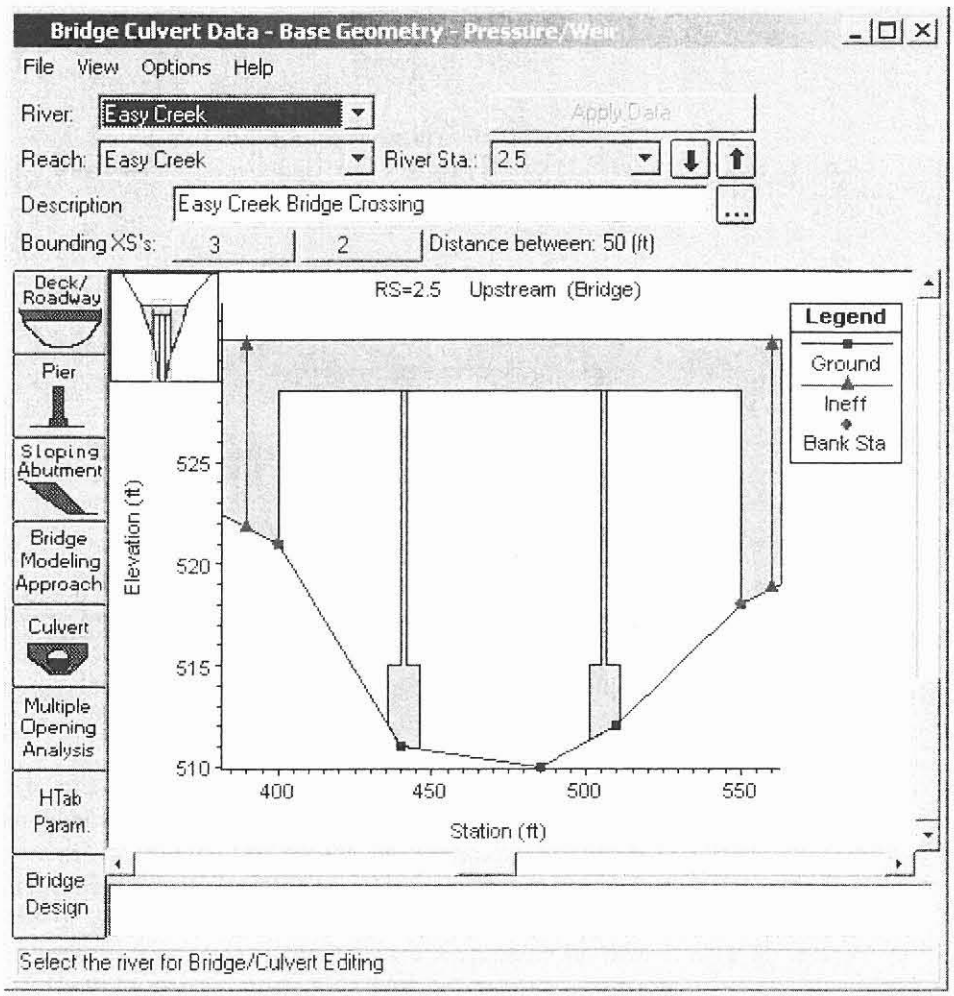


Figure 6.19 Bridge with Piers, zoomed in view

Bridge Modeling Approach. The Bridge Modeling Approach editor is used to define how the bridge will be modeled and to enter any coefficients that are necessary. To bring up the Bridge Modeling Approach editor press the **Bridge Modeling Approach** button on the Bridge/Culvert Data editor. Once this button is pressed, the editor will appear as shown in Figure 6.20 (except yours will only have the default methods selected).

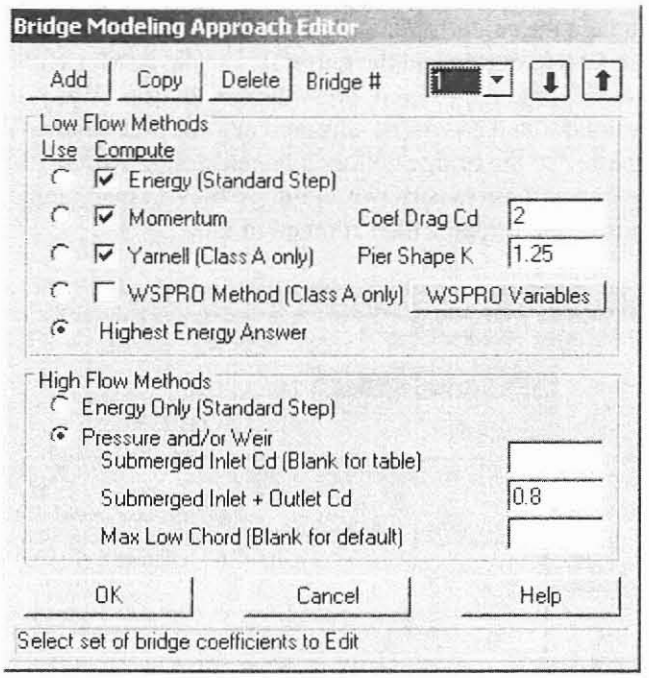


Figure 6.20 Bridge Modeling Approach Editor

When the Bridge Modeling Approach editor comes up it will be ready to enter data for the first bridge opening (coefficient set # 1). If there is more than one bridge opening at the current location, the user can either use a single set of modeling approaches and coefficients, or establish a different set for each bridge opening.

Establishing a bridge modeling approach consists of defining which methods the program will use for low flow computations and high flow (flow at or above the maximum low chord) computations. The user can instruct the program to use any or all of the low flow methods during the computations by clicking the buttons under the **Compute** column. If either the Momentum or Yarnell method are selected, the user must enter a value for the pier loss coefficient that corresponds to that method. If the WSPRO method is selected, the user must press the "WSPRO Variables" button and enter additional information that is required for the method. Once the **WSPRO Variables** button is pressed, a data editor as shown in Figure 6.21 will appear.

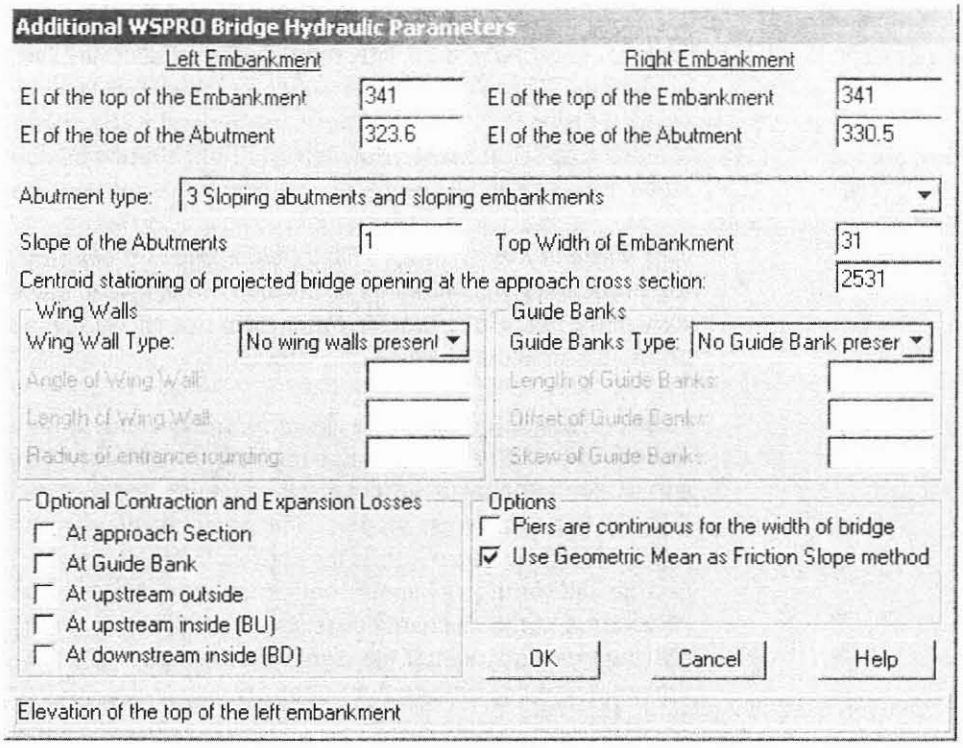


Figure 6.21 WSPRO Data Editor

As shown in Figure 6.21, there are several variables that must be entered as well as some options that are available to the user. All of the required variables shown on the WSPRO data editor are used in the computation of the discharge coefficient, C, which is used in the WSPRO expansion loss equation. A detailed discussion of how the discharge coefficient is computed can be found in appendix D of the HEC-RAS Hydraulic Reference manual. The following is a description of each of the variables on the WSPRO Data Editor:

El of the top of the Embankment - These fields are used for entering the elevation of the top of the embankment (top of road) at the edges of the bridge opening. An elevation must be entered for both the left and right side of the bridge opening.

El of the toe of the Abutment - These fields are used for entering the elevation of the abutment toe (elevation at the station in which the abutment toe intersects with the natural ground inside the bridge opening) on both the left and right side of the bridge opening.

Abutment Type - This field is used for selecting the type of abutments. There are four abutment types available from this selection box.

Slope of the Abutments - This field is used for entering the slope of the abutments. This slope is taken as the horizontal distance divided by the vertical distance. If the abutments are vertical walls, then this field should be left blank or set to zero. If the left and right abutments do not have the same slope, take an average of the two and enter that into this field.

Top Width of Embankment - This field is used for entering the width of the top of the road embankment, in the area of the bridge opening. If the topwidth of the embankment varies from one end of the bridge opening to the other, use an average of the two widths.

Centroid stationing of the projected bridge opening at the approach cross section - For the WSPRO bridge method, it is necessary to calculate the water surface topwidth inside of the bridge opening, and then project that width onto the approach cross section. The program calculates the conveyance within this projected width at the approach cross section. This conveyance is used in calculating a channel contraction ratio, which is an integral part in the calculation of the discharge coefficient. If this field is left blank, the program will automatically center the computed topwidth, such that the center of the topwidth will be at the center of conveyance at the approach cross-section. The user can override this by entering their own centroid stationing value for the approach cross section.

Wing Walls - This field is used for selecting the type of wing walls. There are three choices available in the selection box: No wing walls present; Angular wing walls; and Rounded wing walls. If the user selects "Angular wing walls", then the fields labeled "Angle of Wing Wall" and "Length of Wing Wall" become active and must be filled out. If the user selects "Rounded wing walls", then the fields "Length of wing walls" and "Radius of entrance rounding" become active and must be filled out. If the user selects "No wing walls present" then no other information on wing walls is necessary. For more information on wing walls see appendix D of the HEC-RAS Hydraulic Reference manual.

Guide Banks Type - This field is used for selecting the type of guide banks if any exist. There are three choices available from the selection box: No guide bank present; Straight; and Elliptical. If the user selects "Straight" then the fields labeled "Length of guide banks", "Offset of Guide Banks", and "Skew of Guide Banks" become active and must be filled out. If the use selects "Elliptical" then only the fields "Length of Guide Banks" and "Offset of Guide Banks" become active. If the user selects "No Guide Bank present" then no other information about guide banks is necessary. For more information on Guide Banks see appendix D of the HEC-RAS Hydraulic Reference manual.

Optional Contraction and Expansion Losses - This box allows the user to turn on contraction and expansion losses at locations that are traditionally not in the WSPRO methodology. The basic WSPRO bridge method only computes expansion losses in the expansion reach (between the exit cross section and the section just downstream of the bridge). This option allows the user to turn on contraction and expansion losses individually at the following locations: downstream inside of the bridge; upstream inside of the bridge; upstream outside of the bridge; at the end of a guide bank (if guide banks exist); and at the approach cross section. The default for the WSPRO method is that contraction and expansion losses will not be calculated at these locations. Users should not turn these options on unless they feel that the standard WSPRO bridge approach is not producing enough energy loss through the bridge.

Two other options that the user has control over are: specifying that the piers are continuous the whole way through the bridge or not, and using the Geometric Mean friction slope averaging technique through the bridge computations (from exit to approach section). The default for the WSPRO methodology is to assume that the piers are continuous through the bridge, and to use the Geometric Mean friction slope method.

After all of the variables have been entered, the user must press the **OK** button for the WSPRO variables to be accepted. For more information about the computation of the discharge coefficient, and these data variable, see appendix D of the HEC-RAS Hydraulic Reference manual.

Once the user has selected which low flow bridge methods will be computed, they must also specify which of those methods will be used as the final answer to continue the computations on upstream with. Only one of the methods can be selected as the answer to "**Use**" in order to continue the computations upstream. An alternative to selecting a single method to use is to instruct the program to use the answer with the highest computed upstream energy elevation. This is accomplished by pressing the button under the **"Use"** column that corresponds to the **Highest Energy Answer** text field.

For high flows, the modeler can choose between Energy based calculations or pressure and weir flow calculations. If pressure and weir flow is the selected high flow method, the user must enter coefficients for the pressure flow equations. The first coefficient applies to the equation that is used when only the upstream side (inlet) of the bridge is submerged. If this coefficient is left blank, the program selects a coefficient based on the amount of submergence. If the user enters a coefficient, then that value is used for all degrees of submergence. The second coefficient applies to the equation that is used when both the upstream and downstream end of the bridge is sub-merged. Generally this coefficient is around 0.8. For more information on pressure flow coefficients see Hydraulics of Bridge Waterways (FHWA, 1978).

Max Low Chord - This field is used to set the maximum elevation of the deck low chord, and therefore the elevation at which pressure flow begins to be calculated. If this field is left blank, then the elevation that triggers pressure

flow calculations is based on the highest low chord elevation on the upstream side of the bridge deck. If the user enters a value in this field, then the value set will be used to trigger when pressure flow calculations begin. Pressure flow is triggered when the energy elevation exceeds the maximum low chord. When pressure flow is calculated, the answer is compared to the low flow answer and the higher of the two is selected. Alternatively, the user can tell the program to use the water surface instead of the energy elevation to trigger pressure flow calculations. This option can be found under the **Bridge and Culvert Options** section of this manual.

Once all of the bridge modeling approach information is entered, the user should press the **OK** button. When the OK button is pressed the information will be accepted and the editor will close. **Remember! The data are not saved to disk at this point, it is only accepted as being valid.** To save the geometric data, use the **File** menu from the Geometric Data Editor window.

Bridge Design Editor

The bridge design editor allows the user to enter or modify bridge data quickly and conveniently. With this editor the user can enter the deck/roadway data, sloping abutments, and pier information. To put together a bridge with this editor, the user would do the following:

1. From the Geometric Data window, open the Bridge/Culvert data editor. Select the River and Reach in which you would like to place the bridge.

2. Go to the **Options** menu and select **Add a Bridge and/or Culvert** from the list. An input box will appear prompting you to enter a river station identifier for the new bridge.

3. Open the Bridge Design editor by pressing the **Bridge Design** button on the lower left side of the Bridge/Culvert Data editor.

4. Enter the required data for the bridge deck/roadway, sloping abutments (optional), and piers (optional).

When the **Bridge Design** button is pressed, a window will appear as shown in Figure 6.22. The user only has to enter a minimal amount of information to build or edit the bridge. To create the bridge deck/roadway, the user must enter a high cord elevation (top of road) and a low cord elevation (maximum elevation inside of the bridge opening).

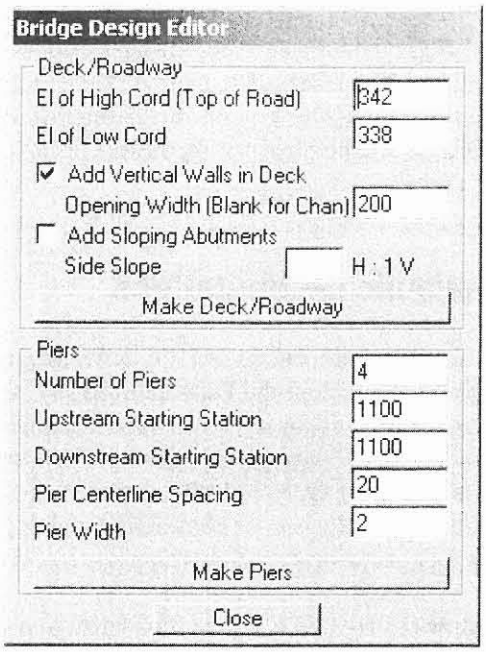


Figure 6.22 Bridge Design Editor

The user has the option to limit the width of the bridge opening by selecting the **Add Vertical Walls in Deck** option. When this option is selected, the bridge opening will be limited to either the main channel bank stations (this is the default) or a user specified width (this is optional). Everything left and right of the bridge opening will be completely filled in all the way to the ground elevations. If the user enters a bridge opening width, the opening will be centered between the main channel bank stations.

The user also has the option to enter sloping abutments. Sloping abutments should only be entered after selecting to limit the width of the bridge opening with the vertical walls option. To enter sloping abutments, the user only has to enter a slope in units of horizontal to vertical. The program will automatically build a left and right sloping abutment that starts in the upper left and right corners of the bridge opening.

Once all of the bridge deck/roadway information is entered, the user can have the program build the deck/roadway by pressing the **Make Deck/Roadway** button.

The last option available in the Bridge Design editor is to enter pier information. The user enters the number of piers, the upstream and downstream stationing of the left most pier, the spacing between the centerline of the piers, and the width of the piers. The user then presses the **Make Piers** button to have the interface build the piers.

After all of the bridge data are entered, the user presses the **Close** button to get out of the editor. The bridge data can be changed at any time by either going back into the Bridge Design editor and entering new values, or by going to the more detailed editors for the bridge deck/roadway, sloping abutments, and piers.

Culvert Hydraulic Computations

The culvert hydraulic computations in HEC-RAS are similar to the bridge hydraulic computations, except the Federal Highway Administration's (FHWA) standard equations for culvert hydraulics under inlet control are used to compute the losses through the structure. Because of the similarities between culverts and other types of bridges, the cross section layout, the use of ineffective areas, the selection of contraction and expansion coefficients, and many other aspects of bridge analysis apply to culverts as well.

The culvert routines in HEC-RAS have the ability to model nine different types of culvert shapes. These shapes include box (rectangular), circular, elliptical, arch, pipe arch, semi circular, low profile arch, high profile arch, and Con Span culverts.

The analysis of flow in culverts is complicated. It is common to use the concepts of "Inlet" control and "Outlet" control to simplify the analysis. **Inlet control** flow occurs when the flow carrying capacity of the culvert entrance is less than the flow capacity of the culvert barrel. **Outlet control** flow occurs when the culvert carrying capacity is limited by downstream conditions or by the flow capacity of the culvert barrel. The HEC-RAS culvert routines compute the headwater required to produce a given flow rate through the culvert for inlet control conditions and for outlet control conditions. In general, the higher headwater "controls," and an upstream water surface is computed to correspond to that energy elevation.

Inlet Control Computations. For inlet control, the required headwater is computed by assuming that the culvert inlet acts as an orifice or a weir. Therefore, the inlet control capacity depends primarily on the geometry of the culvert entrance. Extensive laboratory tests by the National Bureau of Standards, and the Bureau of Public Roads (now, FHWA), and other entities resulted in a series of equations which describe the inlet control headwater under various conditions. These equations are used by HEC-RAS in computing the headwater associated with inlet control.

Outlet Control Computations. For outlet control flow, the required headwater must be computed considering several conditions within the culvert and the downstream tailwater. For culverts flowing full, the total energy loss through the culvert is computed as the sum of friction losses, entrance losses, and exit losses. Friction losses are based on Manning's equation. Entrance losses are computed as a coefficient times the velocity head in the culvert at the upstream end. Exit losses are computed as a coefficient times the change in velocity head from just inside the culvert (at

the downstream end) to outside the culvert.

When the culvert is not flowing full, the direct step backwater procedure is used to calculate the profile through the culvert up to the culvert inlet. An entrance loss is then computed and added to the energy inside the culvert (at the upstream end) to obtain the upstream energy (headwater). For more information on the hydraulics of culverts, the reader is referred to Chapter 6 of the HEC-RAS Hydraulics Reference manual.

Entering and Editing Culvert Data

Culvert data are entered in the same manner as bridge data. To enter culvert data the user presses the **Bridge/Culvert** button on the Geometric Data window (Figure 6.1). Once this button is pressed, the Bridge/Culvert Data Editor will appear (Figure 6.12). To add a culvert group to the model the user must then do the following:

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

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

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

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

River, Reach and River Station. The River and Reach boxes allow the user to select a river and reach from the available reaches that were put together in the schematic diagram. The reach label defines which reach the culvert will be located in. The River Station tag defines where the culvert will be located within the specified reach. The River Station tag does not have to be the actual river station of the culvert, but it must be a numeric value. The River Station tag for the culvert should be numerically between the two cross sections that bound the culvert. Once the user selects **Add a Bridge and/or Culvert** from the options menu, an input box will appear prompting you to enter a River Station tag for the new culvert. After the River Station tag is

entered, the two cross sections that bound the culvert will be displayed on the editor.

Description. The description box is used to describe the culvert location in more detail than just the river, reach and river station. This box has a limit of 256 characters. Only the first line of information is displayed, unless the button to the right of the box is pressed. Also, the first 40 characters of the description are used as a label for culvert plots and tables.

Culvert Road Embankment. The culvert road embankment is virtually the same as the bridge deck/roadway information. The road embankment is used to describe the area blocking the stream and the roadway profile. The only difference in the information for culverts is that the low chord elevations should be left blank or set to elevations below the ground data. This will cause the road embankment to completely fill the channel up to the roadway elevations (high chord data). Therefore, the only opening below the roadway will be whatever culvert openings are entered.

To enter the culvert roadway information, press the **Deck/Roadway** button on the Bridge/Culvert Data Editor window. For an explanation of the deck information, please review the section entitled **Bridge Deck/Roadway** found earlier in this chapter.

Culvert Data. To enter culvert specific information, press the **Culvert** button on the Bridge/Culvert Data Editor window. When this button is pressed, the Culvert Data Editor will appear as shown in Figure 6.23 (except yours will be blank). The information entered in the Culvert Data Editor consists of the following:

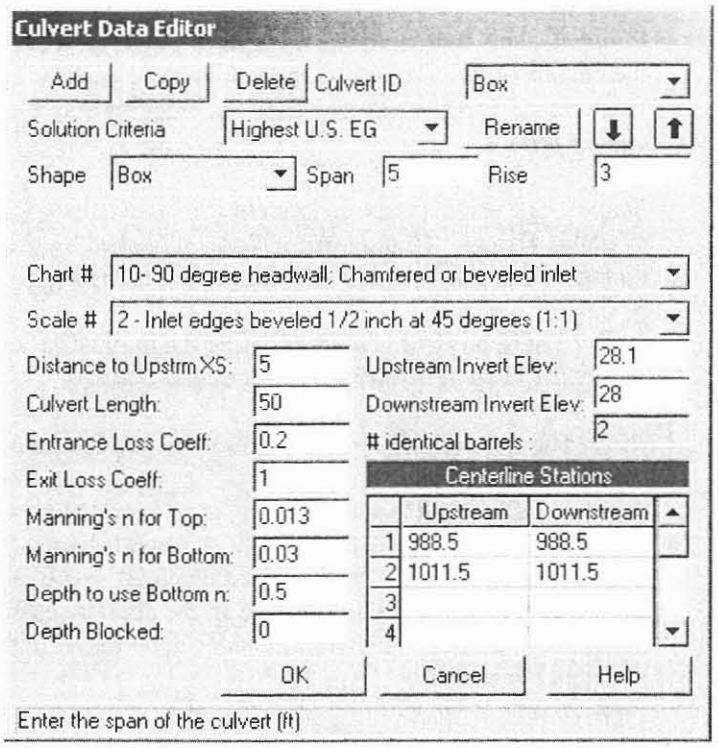


Figure 6.23 Culvert Data Editor

Culvert ID# - The culvert identifier (ID#) is automatically assigned to "Culvert #1" the first time you open the editor. The user can enter up to ten culvert types if they are working on a multiple culvert, opening problem. If all of the culvert barrels are exactly the same, then only one culvert type (Culvert ID#) should be entered. The number of barrels is an input parameter in the culvert data. If the user has culverts that are different in shape, size, elevation, or loss coefficients, then additional culverts types (Culvert ID#'s) must be added for each culvert type. To add an additional culvert type you can either use the **Add** or **Copy** buttons. The Add button increments the culvert ID# and clears the culvert editor. The Copy button increments the culvert ID# and makes a copy of the original culvert data. Once a copy is made of a culvert, the user can change any of the existing culvert information. Culverts can be deleted by pressing the **Delete** button.

Solution Criteria - This option allows the user to select between taking the higher of the inlet control and outlet control answers (Highest U.S. EG), or specifically selecting the Inlet control or Outlet control answer. The default is to let the program compute both and take the higher of the two. In general this should be left this way. The only time a user should specifically select Inlet control or Outlet control, is when they feel the program is in error by selecting the higher of the two answers.

Rename - This button allows the user to put in their own identifier for each of the culvert types. By default the culvert types will be labeled "Culvert #1," "Culvert #2," and so on. The user can enter up to twelve characters for each culvert type.

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

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

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

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

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

Distance to Upstream XS - This field is used to locate the culvert in space, relative to the two cross sections that bound the culvert crossing. The user should enter the distance between the upstream cross section and the upstream end of the culvert barrel.

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

Entrance Loss Coefficient - The coefficient entered in this field will be multiplied by the velocity head inside of the culvert at the upstream end. This value represents the amount of energy loss that occurs as flow transitions from the upstream cross section to inside the culvert barrel. This coefficient is used in the outlet control computations.

Exit Loss Coefficient - The coefficient entered in this field will be multiplied by the change in velocity head from inside the culvert to outside the culvert at the downstream end. This value represents the energy loss that occurs as water exits the culvert. This coefficient is used in the outlet control computations.

Manning's n for Top - The n-value fields are used for entering the Manning's n values of the culvert barrel. This version of HEC-RAS allows the user to enter a separate n value for the top (which includes top and sides) of the

culvert, as well as for the bottom. If the culvert has the same roughness for the top and bottom, the user can enter the value for the top. The Manning's n value for the bottom will automatically be copied from the top field.

Manning's n for Bottom – This field is used to enter a Manning's n value for the bottom of the culvert. This n value will be used up to a user specified depth inside of the culvert. When the water surface gets higher than that depth, a composite Manning's n value is computed based on the bottom and top n values and their corresponding wetted perimeters.

Depth to use Bottom n – This field is used to specify the depth that the "Bottom n value" is applied inside of the culvert. The surface of the culvert below this depth is given the n value for the bottom of the culvert, while the surface of the culvert above this depth is given the n value for the top of the culvert.

Depth Blocked – This field is used to block off a portion of the bottom of the culvert. When a value is entered into this field, the culvert is completely blocked up to the depth specified. This blocked out area persists the whole way through the culvert.

Upstream Invert Elevation - This field is used to describe the elevation of the culvert invert at the upstream end.

Downstream Invert Elevation - This field is used to describe the elevation of the culvert invert at the downstream end.

 # Identical Barrels - This field is used to **display** the number of identical barrels. The number of identical barrels is limited to 25. To enter more than one identical barrel, the user must provide different centerline stationing information for each barrel. As the centerline stationing information is added, the number of identical barrels will automatically change to reflect the number of centerline stations. The user does not enter anything into this field, it is just used to display the number of identical barrels.

Centerline Stations - This table is used to enter the stationing of each culvert barrel. Centerline stations must be provided for both the upstream and downstream side of each culvert barrel.

Once all of the culvert information is entered, the user should press the **OK** button at the bottom of the window. Pressing the **OK** button tells the interface to accept the data and close the window. Once the culvert editor is closed, the graphic of the culvert will appear on the Bridge/Culvert Data editor window. An example culvert with two culvert types and two identical barrels for each culvert type is shown in Figure 6.24. **Note! The data are not saved to the hard disk at this point**. Geometric data can only be saved from the **File** menu on the Geometric Data window.

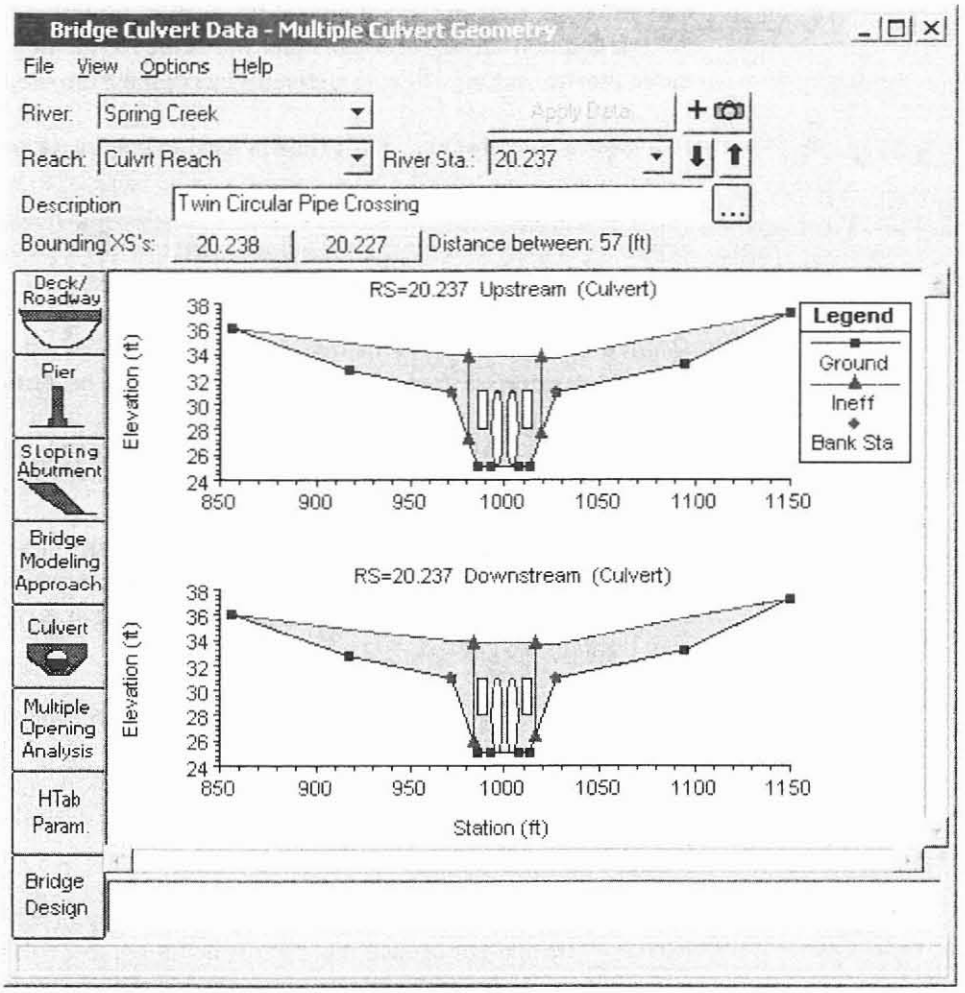


Figure 6.24 Bridge/Culvert Data Editor with example culvert

Bridge and Culvert Options

Some additional options that are available, but not required, are found under the **Options** menu from the Bridge/Culvert Data Editor. These include the following:

Add a Bridge and/or Culvert. This option initiates the process of adding a bridge or culvert to the data set. The user is prompted to enter a river station tag for the new bridge or culvert. The river station tag locates the bridge or culvert within the selected reach. Once the river station is entered, the Bridge/Culvert Data editor is cleared and the user can begin entering the data for that new bridge or culvert.

Copy Bridge and/or Culvert. This option allows the user to make a copy of the bridge and/or culvert crossing and place it in another reach and/or river station within the current project.

Rename River Station. This option allows the user to change the river station of the currently opened Bridge and/or Culvert crossing.

Delete Bridge and/or Culvert. This option will delete the currently displayed bridge or culvert. The user is prompted with a message stating specifically which bridge or culvert is going to be deleted, and requesting them to press the **OK** button or the **Cancel** button.

Internal Bridge Cross-Sections. This option allows the user to edit the two cross sections inside of a bridge. These two cross sections are a copy of the cross sections just upstream and downstream of the bridge. If the ground elevations inside of the bridge are different than just outside of the bridge, then the internal bridge cross sections should be modified to reflect the changing elevations. This option allows the user to change the station and elevation data, roughness coefficients, and main channel bank stations for each of the two internal bridge cross sections.

Momentum Equation. This option allows the user to change the components of the momentum equation. The momentum equation is one of the optional low flow methods in the bridge routines. The default momentum equation includes terms in the equation to account for friction losses and the weight of water component. The user can turn either or both of these components off from this option.

Momentum Class B Defaults. If the program computes that the flow must pass through critical depth inside the bridge (Class B flow), critical depth will automatically be located inside the bridge at the most constricted cross section. If both cross sections are identical, the program will locate critical depth at the upstream inside cross section. This option allows the user to control where the program sets critical depth for class B flow. If the user feels that it would be better to set critical depth inside the bridge at the downstream end, then this can be selected.

Pressure Flow Criteria. This option allows the user to select either the energy grade line or the water surface, to be used as the criterion for when the program begins checking for the possibility of pressure flow. By default the program uses the energy grade line. This does not change how pressure flow is calculated, only when the program will begin checking for pressure flow.

Ice Option. This option allows the user to select how ice will be handled inside of the bridge during ice computations. This option is only pertinent if the user is performing a profile computation with the effects of ice included. When this option is selected, a window will appear asking the user to select one of three available options. These options include: no ice inside of the bridge; a constant amount of ice through the bridge; dynamic ice effects are to be computed through the bridge.

Skew Bridge/Culvert. This option allows the user to make adjustments to bridge/culvert data that is skewed (i.e. not perpendicular to the flow lines going through the bridge/culvert. When this option is selected, a window will appear allowing the user to enter a skew angle for the deck/roadway, as well as the piers. The stationing of the deck/roadway is reduced, by multiplying it by the cosine of the user entered skew angle. Additionally, the user has the option to adjust the upstream and downstream cross sections bounding the bridge by the same skew angle. A separate skew angle is entered for bridge piers. The piers are assumed to go the whole way through the bridge as a single continuous pier.

Bridge and Culvert View Features

Several options are available for viewing the bridge/culvert geometric data. These options include: Zoom In; Zoom Out; Display Upstream XS; Display Downstream XS; Display Both; Highlight Weir, Opening Lid and Ground; Highlight Piers; and Grid. These options are available from the **View** menu on the bridge/culvert data editor.

Zoom In. This option allows the user to zoom in on a piece of the bridge or culvert. This is accomplished by selecting **Zoom In** from the **View** menu, then specifying the area to zoom in on with the mouse. Defining the zoom area is accomplished by placing the mouse pointer in the upper left corner of the desired area. Then press down on the left mouse button and drag the mouse to define a box containing the desired zoom area. Finally, release the left mouse button and the viewing area will display the zoomed in area of the bridge or culvert.

Zoom Out. This option displays the bridge or culvert back into its original size before you zoomed in. Zooming out is accomplished by selecting **Zoom Out** from the **View** menu bar on the bridge/culvert data editor.

Full Plot. When this option is selected, the graphic is automatically redrawn back to its full extent, showing the entire bridge/culvert.

Pan. When this option is selected, the user can move the zoomed in portion of the graphic. This is accomplished by first selecting the Pan option, then pressing and holding down the left mouse button while over the graphic. Next, move the graphic in the desired direction, and then release the left mouse button. The graphic will be redrawn with a new portion of the graphic shown in the zoomed in area.

Display Upstream XS. When this option is selected, only the upstream side of the bridge or culvert will be displayed.

Display Downstream XS. When this option is selected, only the downstream side of the bridge or culvert will be displayed.

Display Both. When this option is selected, both the downstream and

upstream sides of the bridge will be displayed in the viewing area.

Highlight Weir, Opening Lid and Ground. When this option is selected, various portions of the bridge/culvert graphic will be highlighted. The program will highlight in red the combination of the deck/roadway high cord and any ground to the left and right of this data. The red color shows what the program will use for weir flow if the Pressure and Weir option is selected for high flows.

The program will also highlight any bridge openings. Within the bridge opening, the ground information will be highlighted in blue and the lid of the opening (deck/roadway low cord data) will be highlighted in green. If the any of these three colors show up in an area where they should not be, then there must be a geometric mistake in the data. This option is very useful for detecting any data entry errors that may otherwise go unnoticed.

Highlight Piers. When this option is turned on the interface will highlight what it thinks is the extent of the pier information. This option allows the user to see exactly what the program thinks piers are, and to see how the pier information has been clipped. Piers are clipped below the ground and above the low chord of the bridge.

Grid. This option allows the user to have a grid overlaid on top of the bridge or culvert graphic.

Multiple Bridge and/or Culvert Openings

HEC-RAS has the ability to model multiple bridge and/or culvert openings at any individual river crossing. Types of openings can consist of bridges, culvert groups (a group of culverts is considered to be a single opening), and conveyance areas (an area where water will flow as open channel flow, other than a bridge or culvert opening). Up to seven openings can be modeled at a given location, and any combination of bridges and culvert groups can be used. Conveyance type openings can range from zero to a maximum of two, and the conveyance areas must be located on the far left and far right of the river crossing.

An example multiple opening is shown in Figure 6.25. As shown in this example, there are three types of openings: a conveyance area (left side, labeled as opening #1), a bridge (labeled as opening #2), and a culvert group (labeled as opening #3). During low flow conditions, flow will be limited to the bridge opening. As flow increases, the culverts will begin to take some of the flow away from the bridge opening. The conveyance area was defined as ineffective flow (no conveyance) until the water surface goes above the top of the bridge. This was accomplished by setting blocked ineffective flow areas. In this example, three blocked ineffective flow areas were established: one to the left of the bridge (which encompasses the whole conveyance area), one between the bridge and the culvert group, and one to the right of the culvert group.

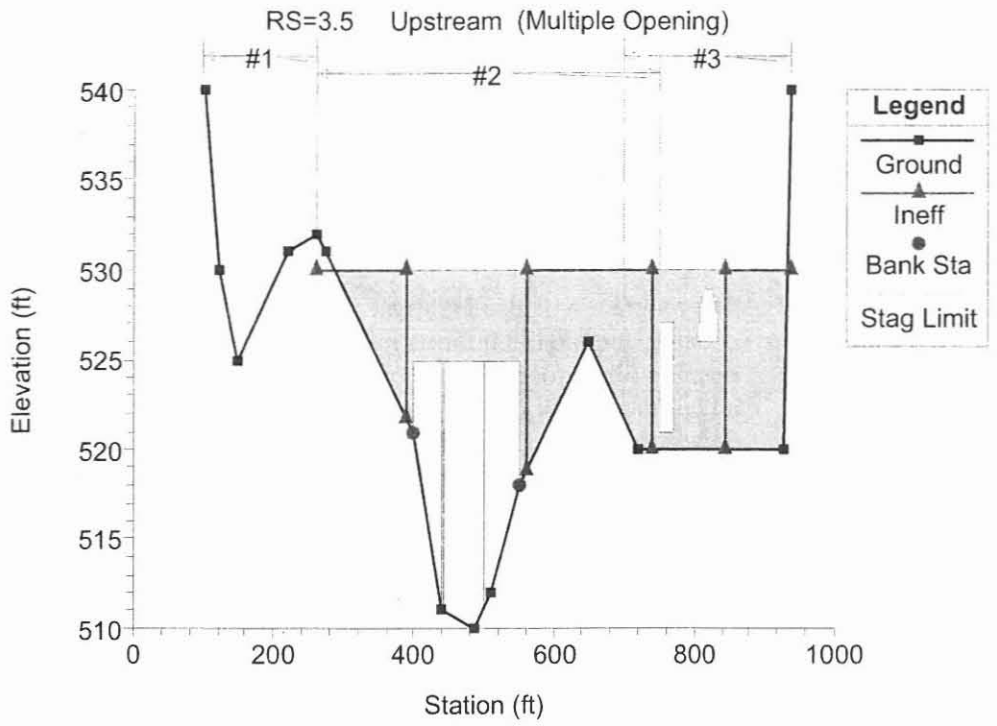


Figure 6.25 Example Multiple Opening River Crossing

Entering Multiple Opening Data

Multiple opening data are entered in the same manner as any other bridge or culvert crossing. In general, the user should perform the following steps to enter multiple opening data:

1. Press the Bridge/Culvert button on the Geometric Data window.

2. Select the river and reach in which you would like to place the multiple opening river crossing. This is accomplished from the River and Reach boxes near the top of the window.

3. Select **Add a Bridge and/or Culvert** from the **Options** menu of the bridge and culvert editor. Enter the river station at which you want to place the multiple opening crossing. Once you have done this, the two cross sections that bound this river station will appear in the window. These two cross sections, along with the bridge and culvert information, will be used to formulate the two cross sections inside the multiple opening river crossing.

4. Enter the deck and road embankment data by using the Deck/Roadway editor.

5. Enter any piers or sloping abutments that are required.

6. Select the **Bridge Modeling Approach** button and enter a set of coefficients and modeling approaches for each bridge opening.

7. Enter Culvert data for any culvert openings.

8. Select the **Multiple Opening Analysis** button on the bridge and culvert editor. Enter the types of openings and their station limits. Start at the left most station of the crossing and work your way to the right end. This is explained in greater detail under the section entitled "Defining the Openings".

Deck/Road Embankment Data. There can only be one deck and road embankment entered for any bridge and/or culvert crossing. The deck editor is used to describe the area that will be blocked out due to the bridge deck and road embankment. As shown by the gray shaded area in Figure 6.25, the deck and roadway data are used to block out area around the bridge as well as around the culverts. In the area of the bridge, high and low chord information is entered in order to define the top of road as well as the bridge opening. In the area of the culverts, the high chord information is entered to define the rest of the top of the road embankment. However, the low chord information can be left blank, or set to elevations below the ground, because the culvert data define the culvert openings.

Piers and abutments. All piers are entered from the pier editor, which was described previously under bridge data. The number of bridge openings has no impact on how pier data are entered. Piers are treated as separate information. Once the user establishes that there is more than one bridge opening, the program is smart enough to figure out which piers go with which opening. If any sloping abutment data are required for a bridge opening, it can be entered as described previously under the bridge data section.

Bridge Modeling Approach. A bridge modeling approach and coefficient set must be established for at least one bridge opening. If there is more than one bridge opening, and the user has only established a single coefficient set and bridge modeling approach, those data will be used for all of the bridge openings. The user can establish a different set of coefficients and modeling approaches for each bridge opening.

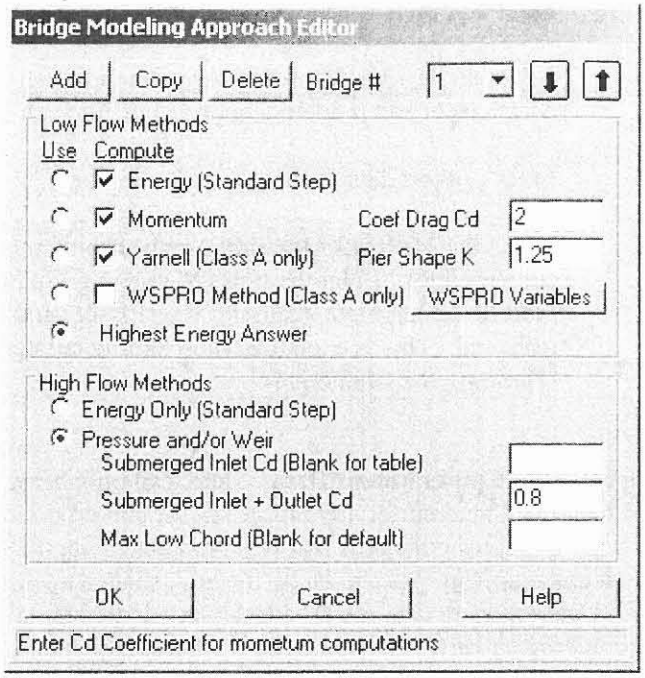


Figure 6.26 Bridge Modeling Approach Editor

As shown in Figure 6.26, the user must enter information under the Bridge Modeling Approach editor for at least one bridge Opening. Bridge openings are referred to as Bridge # 1, Bridge # 2, etc., up to the number of bridge openings. Bridge # 1 represents the left most bridge opening while looking in the downstream direction. Bridge # 2 represents the next bridge opening to the right of Bridge # 1, and so on. The user can enter additional coefficient sets and modeling approaches by selecting either the **Add** or **Copy** button. If either of these buttons is selected, the Bridge # will automatically be incremented by one. The user can then enter or change any of the information on the editor for the second bridge opening. Any bridge opening that does not have a corresponding coefficient set and modeling approach, will automatically default to what is set for Bridge # 1.

Culvert Data. Culvert information is added in the same manner as described in the previous section called "Entering and Editing Culvert Data." Culverts will automatically be grouped based on their stationing.

Defining The Openings

Once all of the bridge and/or culvert data are entered for a multiple opening river crossing, the last step is to define the number and type of openings that are being modeled. This is accomplished by pressing the **Multiple Opening Analysis** button on the Bridge/Culvert Data editor. Once this button is pressed, an editor will appear as shown in Figure 6.27 (except yours will be blank the first time you bring it up).

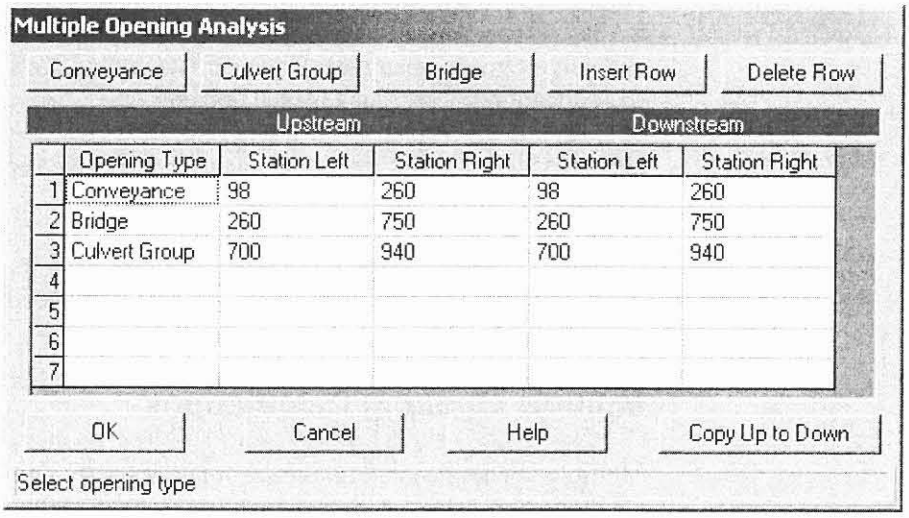


Figure 6.27 Multiple Opening Analysis window

The user selects from the three available opening types: Conveyance; Culvert Group; and Bridge. Openings must be established in order from left to right, while looking in the downstream direction. In addition to establishing the number and types of openings, the user must also enter a Station Left and a Station Right for each opening. These stations are used to establish limits for each opening as well as stagnation points. Stagnation points are the locations at which flow separates (on the upstream side) from one opening to the next adjacent opening. Stagnation points can either be set to fixed locations or they can be allowed to migrate within limits.

As shown in Figure 6.27 (numerical representation) and Figure 6.25 (graphical representation), there are three openings established in this example. The first opening is defined as a conveyance area, and it ranges from station 98 (the left most station of the section) to station 260. That means that any water in this area will be treated as normal open channel flow, and the water surface will be calculated by performing standard step calculations with the energy equation. The second opening is the bridge opening. This opening has a left station of 260 and a right station of 740. This bridge will be modeled by using the cross section data, bridge deck, and pier information that lie within these two stations (260 and 740). The bridge coefficients and modeling approach for this opening will be based on the data

entered for bridge opening #1, since it is the first bridge opening. The third opening is a culvert group. This opening has a left station of 650 and a right station of 940. Any culverts that lie within these stations will be considered as being in the same culvert group.

Notice that the right station of the bridge opening overlaps with the left station of the culvert group. This is done on purpose. By overlapping these stations, the user is allowing the program to calculate the location of the stagnation point between these two openings. This allows the stagnation point to vary from one profile to the next. In the current version of the HEC-RAS software, stagnation points are allowed to migrate between any bridge and culvert group openings. However, stagnation points must be set to a fixed location for any conveyance opening type. A more detailed explanation of stagnation points, and how the program uses them, can be found in the HEC-RAS Hydraulics Reference manual, under the section on Multiple Openings (Chapter 7).

Once the user has entered all of the information into the Multiple Opening Analysis window, simply press the **OK** button to accept the data.

Multiple Opening Calculations

Multiple opening calculations are computationally intensive. An iterative solution approach is used, by which the amount of flow through each opening is adjusted until the computed upstream energies of each opening are balanced within a predefined tolerance. The general approach of the solution scheme is as follows:

1. The program makes a first guess at the upstream water surface by setting it to the computed energy of the cross section just downstream of the bridge.

2. The program sets an initial flow distribution. This is accomplished by first calculating the amount of active flow area in each opening, based on the water surface from step one. The program then apportions the flow by using an area weighting (i.e., if an opening has 40 percent of the active flow area, then it will receive 40 percent of the flow).

3. Once a flow distribution is established, the program then calculates the water surface and energy profiles for each opening, using the estimated flow.

4. Once the program has computed the upstream energy for each opening, a comparison is made between the energies to see if a balance has been achieved (i.e., all energies are within the predefined tolerance). If the energies are not within the set tolerance, the program re-distributes the flow based on the computed energies.

5. The program continues this process until either the computed energies are within the tolerance or the number of iterations reaches a pre-defined maximum. The energy balance tolerance is set as 3 times the user entered water surface calculation tolerance (The default is 0.03 feet or 0.009 meters). The maximum number of iterations for multiple opening analysis is set to 1.5 times the user entered maximum number of iterations from the normal water surface calculations (the default is 30 for multiple openings).

A more detailed discussion of how the program performs the multiple opening analyses can be found in Chapter 7 of the HEC-RAS Hydraulic Reference manual.

Inline Structures (Weirs and Gated Spillways)

HEC-RAS has the ability to model inline weirs and gated structures with radial gates (often called tainter gates) or vertical lift gates (sluice gates). The spillway crest of the gates can be modeled as either an ogee shape or a broad crested weir shape.

This section of the User's manual will describe how to enter the data for inline structures. For information on general modeling guidelines and the hydraulic computations of inline weirs and gated spillways, please see Chapter 8 of the HEC-RAS Hydraulic Reference manual. To find out how to view specific results for an inline structure, see Chapter 9 of this User's manual.

Entering and Editing Inline Structure Data

Inline structure data are entered in a similar manner as bridge and culvert data. To enter an inline weir and/or gated spillway press the **Inline Structure** button from the Geometric Data window. Once this button is pressed, the Inline Structure Data editor will appear as shown in Figure 6.28 (except yours will be blank until you have entered some data).

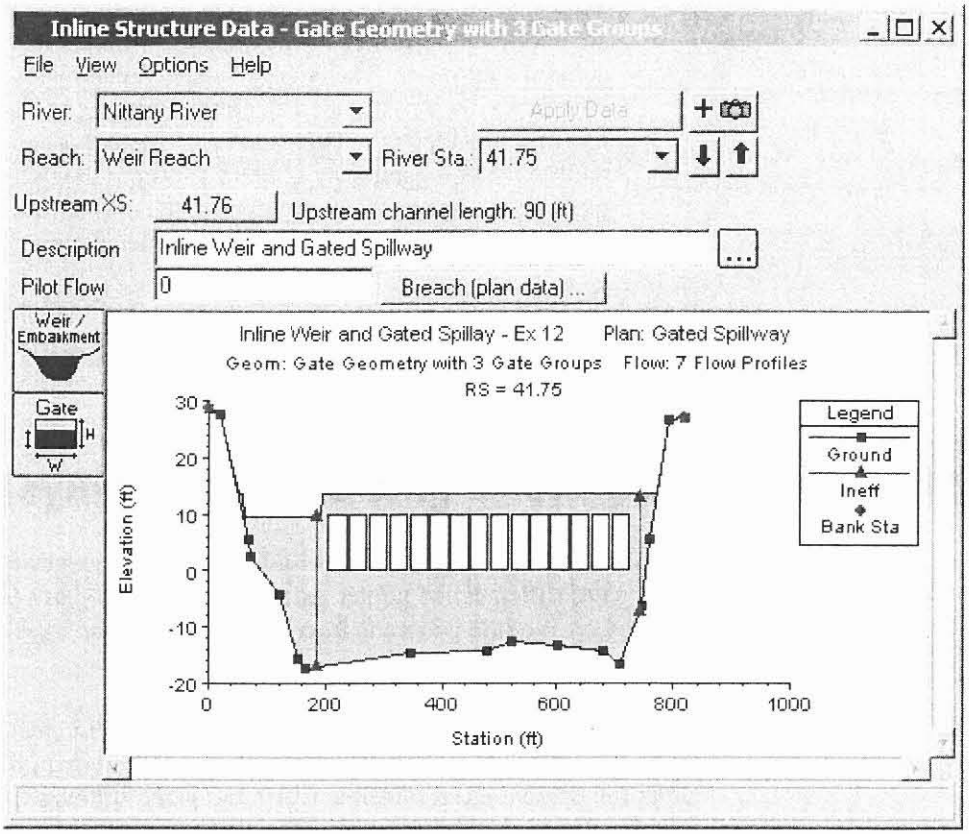


Figure 6.28 Inline Structure Data Editor

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

1. Select the river and reach that you would like to place this inline structure into. This is accomplished by first selecting a River, then selecting a specific reach within that river. The River and Reach selection buttons are at the top of the Inline Structure Data editor.

2. Go to the **Options** menu at the top of the window and select **Add an Inline Structure** from the list. An input box will appear asking you to enter a river station identifier for locating this structure within the reach. After entering the river station, press the **OK** button and a copy of the cross section just upstream of this river station will appear on the screen. This cross section is used in formulating the inline structure crossing.

3. Enter all of the data for the Inline structure. This data will include a Weir/Embankment profile, and any gated spillways that you may be modeling. Gated spillways are optional. If the user does not enter any gated spillways, then the program assumes that there is only an inline weir.

4. Once all of the Inline Structure data are entered, press the **Apply Data** button in order for the interface to accept the data. The editor can then be closed by selecting **Exit** from the **File** menu at the top of the window.

River, Reach, and River Station. The River and Reach boxes allow the user to select a river and reach from the available reaches that were put together in the schematic diagram. The river and reach labels define which river and reach the inline structure will be located in. The River Station tag defines where the structure will be located within the specified reach. The River Station tag does not have to be the actual river station of the structure, but it must be a numeric value. The River Station tag for the inline structure should be numerically between the two cross sections that bound the structure. Once the user selects **Add an Inline Structure** from the options menu, an input box will appear prompting you to enter a River Station tag for the new structure. After the River Station tag is entered, the cross section just upstream of the Inline Structure will be displayed on the editor.

Description. The description box is used to describe the Inline Structure location in more detail than just the river, reach and river station. This box has a limit of 256 characters. Only the first line of information is displayed, unless the button to the right of the box is pressed. Also, the first 40 characters of the description are used as a label for the Inline Structure plots and tables.

Pilot Flow. This option allows the user to put in a flow rate that will be used as a minimum flow release from the structure. If you have an inline structure in HEC-RAS, there must be flow coming out of the structure at all times. The pilot flow option is a simple way to ensure that there is always some flow going through the structure.

Breach (Plan Data). This button allows the user to define information for evaluating the breaching of this inline structure. The data is actually stored in the currently opened plan file. The editor can also be brought up from the plan editor. This option is only for unsteady flow modeling. To learn more about this option, see Chapter 8 "Performing an Unsteady Flow Analysis."

Weir/Embankment. The Embankment and Weir data are entered together, and are used to describe the embankment blocking the stream as well as any uncontrolled weirs. To enter the weir and embankment data, press the **Weir/Embankment** button and the editor will appear (Figure 6-29). The Weir/Embankment Data editor is similar to the Deck/Roadway editor for bridges and culverts. The data on the Weir/Embankment editor is the following:

Distance - The distance field is used to enter the distance between the upstream side of the Weir/Embankment (the top of the embankment) and the cross section immediately upstream of the structure. This distance is entered in feet (or meters for metric).

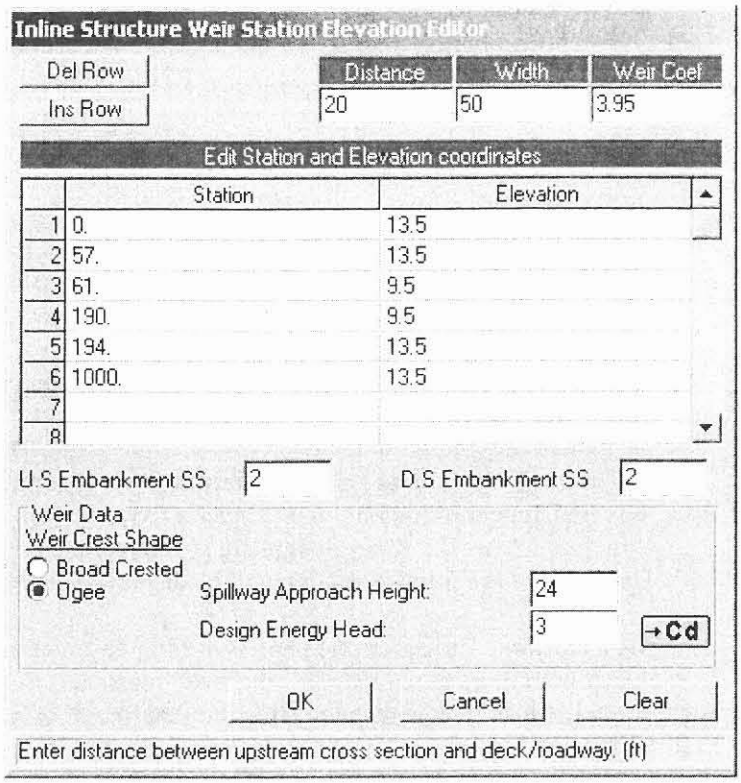


Figure 6.29 Weir and Embankment Data Editor

Width - The width field is used to enter the width of the top of the embankment along the stream. The distance between the top of the downstream side of the embankment and the downstream bounding cross section will equal the main channel reach length of the upstream cross section minus the sum of the weir/embankment "width" and the "distance" between the embankment and the upstream section. The width of the embankment should be entered in feet (meters for metric).

Weir Coefficient - Coefficient that will be used for weir flow over the embankment in the standard weir equation.

Station and Elevation Coordinates - This table is used to define the geometry of the Weir and the Embankment. The information is entered from left to right in cross section stationing. The user enters stations and elevations of the top of the embankment and weir. The stationing does not have to equal the stations in the bounding cross section, but it must be based on the same origin. Everything below these elevations will be filled in down to the ground. The **Del Row** and **Ins Row** buttons allow the user to delete and insert rows.

U.S. Embankment SS - This field is used to enter the slope of the road embankment on the upstream side of the structure. The slope should be entered as the horizontal to vertical distance ratio of the embankment.

D.S. Embankment SS - This field is used to enter the slope of the road embankment on the downstream side of the structure. The slope should be entered as the horizontal to vertical distance ratio of the embankment.

Weir Crest Shape - When submergence occurs over the weir there are two choices available to figure out how much the weir coefficient should be reduced due to the submergence. These two criteria are based on the shape of the weir. The first method is based on work that was done on a trapezoidal shaped broad crested weir (FHWA, 1978). The second criterion was developed for an Ogee spillway shape (COE, 1965). The user should pick the criterion that best matches their problem. If the user selects the Ogee Spillway shape, then some additional information is required. For an Ogee shaped weir the user must enter the "Spillway Approach Height" and the "Design Energy Head". The spillway approach height is equal to the elevation of the spillway crest minus the mean elevation of the ground just upstream of the spillway. The design energy head is equal to the energy grade line elevation (at the design discharge) minus the elevation of the spillway crest. In addition to these two parameters, the user has the option to have the program calculate the weir coefficient at the design discharge. This is accomplished by pressing the $\mathbf{C_d}$ button. Once this button is pressed, the program will compute a weir coefficient for the Ogee spillway based on the design head. During the weir calculations, this coefficient will fluctuate based on the actual head going over the spillway. The curves used for calculating the Ogee spillway coefficient at design head, and discharges other than design head, were taken from the Bureau of Reclamation publication "Design of Small Dams", Figures 249 and 250 on page 378 (Bureau of Reclamation, 1977).

Gated Spillway Data. In addition to uncontrolled overflow weirs, the user can add gated spillways (this is optional). To add gated spillways to the structure, press the **Gate** button on the Inline Structure data editor. Once this button is pressed, the gated editor will appear as shown in Figure 6.30 (except yours will be blank until you have entered some data).

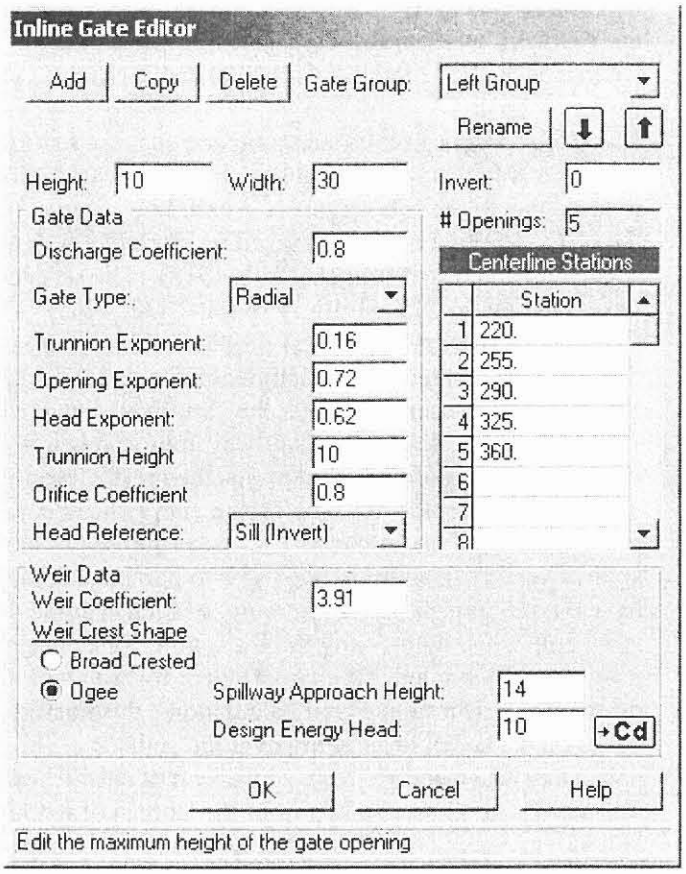


Figure 6.30 Gated Spillway Editor

The Gated Spillway editor is similar to the Culvert editor in concept. The user enters the physical description of the gates, as well as the required coefficients, in the Gated Spillway editor. The functionality of the gates is defined as part of the Steady Flow data, on a per profile basis. The following is a list of the data contained on this editor:

Gate Group - The Gate Group is automatically assigned to "Gate #1" the first time you open the editor. The user can enter up to 10 different Gate Groups at each particular river crossing, and each gate group can have up to 25 identical gate openings. If all of the gate openings are exactly the same, then only one gate group needs to be entered. If the user has gate openings that are different in shape, size, elevation, or have different coefficients, then additional Gate Groups must be added for each Gate type. To add an additional gate group you can either use the **Add** or **Copy** buttons. The Add button increments the Gate # and clears the culvert editor. The Copy button increments the Gate # and makes a copy of the original Gate group data. Once a copy is made of a gate data, the user can change any of the existing gate information. Gate groups can be deleted by pressing the **Delete** button. Also, if the gates are identical, but the user wants to be able to open the gates to different elevations, then the user must have a separate gate group for each set of gates that will be opened to different elevations.

Height - This field is used to enter the maximum possible height that the gate can be opened in feet (meters for metric).

Width - This field is used for entering the width of the gate in feet (meters).

Invert - This field is used for entering the elevation of the gate invert (sill elevation of the spillway inside of the gate) in feet (meters for metric).

Discharge Coefficient - This field is used for entering the coefficient of discharge for the gate opening. This coefficient ranges from 0.6 to 0.8 for Radial gates and 0.5 to 0.7 for sluice gates.

Gate Type - This field is used for selecting the type of gate. Two gate types are available, radial (tainter gate) or sluice (vertical lift gate).

Trunnion Exponent - This field is used to enter the trunnion height exponent, which is used in the radial gate equation. The default value for this field is 0.0.

Opening Exponent - This field is used to enter the gate opening exponent, which is used in the radial gate equation. A default value of 1.0 is automatically set for this field.

Head Exponent - This field is used to enter the upstream energy head exponent, which is used in the radial gate equation. A default value of 0.5 is automatically set for this field.

Trunnion Height - This field is used for entering the height from the spillway crest to the trunnion pivot point. See Chapter 8 of the Hydraulic Reference manual for more details on this variable.

Orifice Coefficient - This field is used to enter an orifice coefficient, which will be used for the gate opening when the gate becomes more than 80 percent submerged. Between 67 percent and 80 percent submerged, the program uses a transition between the fully submerged orifice equation and the free flow equations. When the flow is less than 67 percent submerged, the program uses the free flow gate equations.

Head Reference – This field is used to select the reference point for which the upstream energy head will be computed from. The default is the gate sill (invert), which is normally used when the flow through the gate goes out into a channel. If the gate causes the flow to jet out freely into the atmosphere, then the head reference should be selected as the centerline elevation of the gate opening. If the gate crest is an ogee spillway crest, then the center of the gate opening should be used. Ogee spillway crests are normally designed to follow the shape of water jetting freely into the atmosphere.

Centerline Stations - This table is used for entering the centerline stationing of the identical gate openings. The user should enter a different centerline stationing for each gate opening that is part of the current gate group. All

gate openings within the same gate group are exactly identical in every way, except their centerline stationing. As a user adds new centerline stationing values, the number of identical gates in the group is automatically incremented and displayed in the field labeled "# Openings".

Weir Coefficient - This field is used for entering a weir coefficient that will be used for the gate opening. This coefficient will only be used when the gate is opened to an elevation higher than the upstream water surface elevation. When this occurs, the flow through the gate is calculated as weir flow.

Weir Crest Shape - This parameter allows the user to select between a Broad Crested shape weir and an Ogee shaped weir. Depending on which shape is selected, the program will use a different submergence criteria during the calculation. In addition to the submergence criteria, if the user selects the Ogee shape, the program will bring up two additional data entry fields that must be entered by the user. These fields are used for the Spillway Approach Height and the Design Energy Head, which are explained below. Once these fields are entered, the user should press the button labeled $\mathbf{C_d}$. When this button is pressed, the program will compute a weir coefficient for the Ogee spillway based on the design head. During the weir calculations, this coefficient will fluctuate based on the actual head going over the gated spillway. The curves used for calculating the Ogee spillway coefficient at design head, and discharges other than design head, work taken from the Bureau of Reclamation publication "Design of Small Dams", Figures 249 and 250 on page 378 (Bureau of Reclamation, 1977).

Spillway Approach Height - The spillway approach height is equal to the elevation of the spillway crest minus the mean elevation of the ground just upstream of the spillway.

Design Energy Head - The design energy head is equal to the energy grade line elevation (at the design discharge) minus the elevation of the spillway.

Once all of the data for the gates has been entered, the user needs to press the **OK** button for the data to be accepted. If the user does not want to use the new data, and would like to go back to the original data they had before entering the Gate Editor, press the **Cancel** button. If the user presses the OK button, this does not mean that the data is saved to the hard disk, it is only stored in memory and accepted as being good data. This data is part of the geometry data, and is stored in the geometric data file. The data can be stored to the hard disk by selecting one of the save options from the File menu of the Geometric Data window.

Lateral Structures (Weirs, Gated Spillways, Culverts, and Diversion Rating Curves)

HEC-RAS has the ability to model lateral weirs, gated spillways, culverts, and diversion rating curves. The user can set up a single lateral weir, a weir and separate set of gates, a weir and group of culverts, or any combination of weir, gates, culverts, and a rating curve. The gated spillways can have either radial gates (often called tainter gates) or vertical lift gates (sluice gates). The spillway crest of the gates can be modeled as either an ogee shape or a broad crested weir shape. The culverts can be any of the available shapes from the standard HEC-RAS culvert capability. The diversion rating curve can be used alone, or in conjunction with the other hydraulic outlet types. The rating curve can be used to represent a structure or a particular outlet that could not be modeled with HEC-RAS. Lateral structures can be connected to storage areas or another river reach.

The lateral structure option can also be used to model a levee. In general, the user should end their cross sections at the levee, and then use the later structure option to represent the top of the levee along the stream. The area behind the levee could be represented with either a storage area or another river reach. Water that goes above the levee will be modeled as weir flow. The user also has the option to evaluate levee breaching.

This section of the User's manual will describe how to enter the data for lateral weirs, gated spillways, culverts, and lateral rating curves. For information on general modeling guidelines and the hydraulic computations of lateral weirs, gated spillways, and culverts, please see Chapter 8 of the HEC-RAS Hydraulic Reference manual. To find out how to view specific results for a lateral structure, see Chapter 9 of this User's manual.

Entering and Editing Lateral Structure Data

Lateral weir, gated spillway, and culvert data are entered in a similar manner as bridge and culvert data. To enter a lateral structure, press the **Lateral Structure** button from the Geometric Data window. Once this button is pressed, the Lateral Structure Data editor will appear as shown in Figure 6.31 (except yours will be blank until you have entered some data).

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

1. Select the river and reach that you would like to place this lateral structure into. This is accomplished by first selecting a River, then selecting a specific reach within that river. The River and Reach selection buttons are at the top of the Lateral Structure Data editor.

2. Go to the **Options** menu at the top of the window and select **Add a Lateral Structure** from the list. An input box will appear asking you to enter a river station identifier for locating this structure within the

reach. The river station you enter will represent the location of the upstream end of the lateral structure. The river station must be unique, and should be numerical between the river station values of the upstream cross section and the next section downstream. After entering the river station, press the **OK** button and a profile plot of the channel invert and cross sections in the vicinity of the lateral weir/spillway will be displayed.

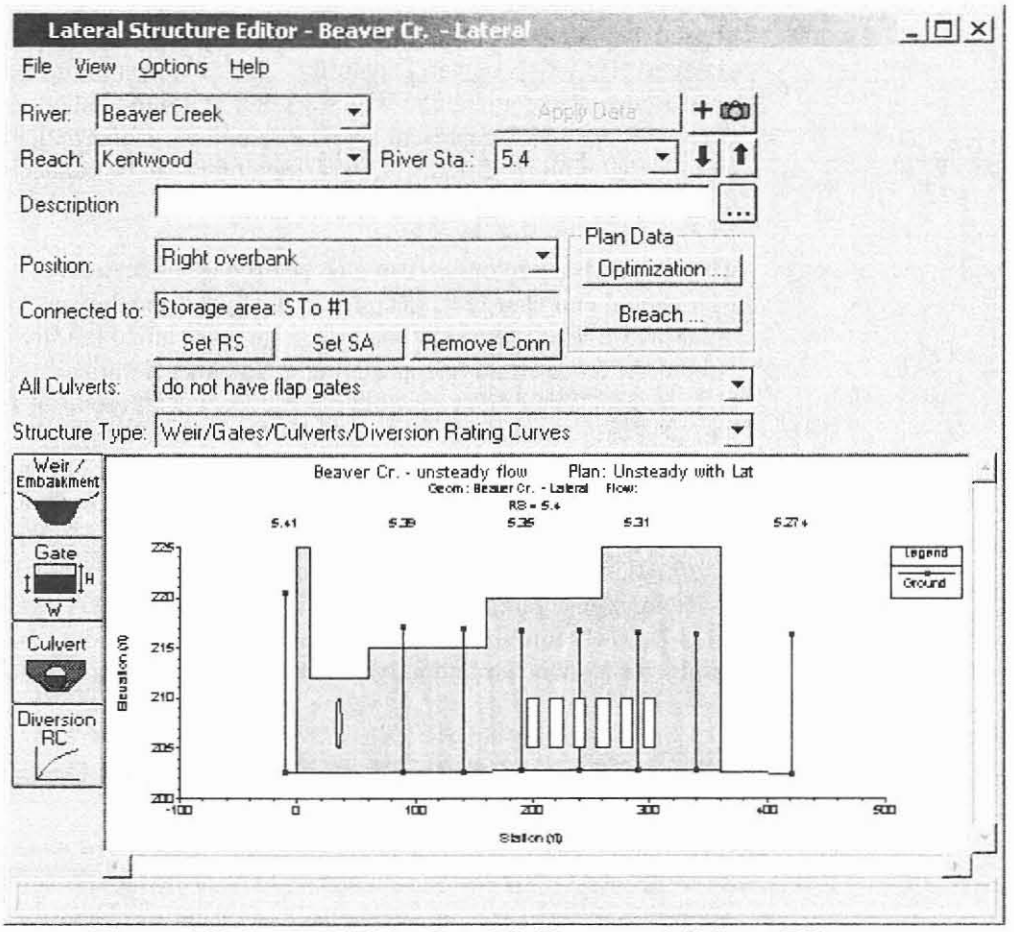


Figure 6.31 Lateral Weir, Gated Spillway, and Culvert Editor

3. Enter all of the data for the Lateral Weir, Gated Spillways, Culverts and Diversion Rating Curves. Gated spillways, culverts, and rating curves are optional. If the user does not enter any gated spillways, culverts, or rating curves, then the program assumes that there is only a lateral weir. If the user wants to enter only gated spillways, culverts, or a rating curve, and no lateral weir, they must still enter a weir embankment.

4. Once all of the data are entered, press the **Apply Data** button in order for the interface to accept the data. The editor can then be closed by selecting **Exit** from the **File** menu at the top of the window.

5. If any gated spillways were entered, the user must go to the Steady or Unsteady Flow Data Editor to control the gate settings for each individual event.

The user can have up to two lateral structures defined between any two given cross sections. However, the lateral structure must be placed on opposite sides of the channel (i.e. one on the left and one on the right), and the river stations of each lateral structure must be different (though still contained within the two cross section river station values). Also, any lateral structure can be longer than the distance between cross sections. The user can have a lateral structure that extends downstream, encompassing up to 100 cross sections. If you have a lateral structure that is longer than that, you must break it up into separate lateral structure.

River, Reach, and River Station. The River and Reach boxes allow the user to select a river and reach from the available reaches that were defined in the schematic diagram. The river and reach labels define which river and reach the lateral weir and/or gate spillway will be located in. The River Station tag defines where the structure will be located within the specified reach. The River Station tag does not have to be the actual river station of the structure, but it must be a numeric value. The River Station tag for the lateral structure should be numerically between the two cross sections that bound the upstream end of the structure. Once the user selects **Add a Lateral Structure** from the options menu, an input box will appear prompting you to enter a River Station tag for the new structure. After the River Station tag is entered, a profile plot of the reach thalweg will be displayed for the bounding cross sections in the graphic window.

Description. The description box is used to describe the Lateral Structure location in more detail than just the river, reach and river station. This box has a limit of 256 characters. Only the first line of information is displayed, unless the button to the right of the box is pressed. Also, the first 40 characters of the description are used as a label for the Lateral Structure plots and tables.

Position. The position box is used to define where the lateral structure is located spatially within the cross section. The user can select one of the following: Left overbank; Next to left bank station; Next to right bank station; and Right overbank. When the user selects "Left overbank", the weir is assumed to be located at the left end (beginning cross section station) of the cross section data, looking in the downstream direction. When the user selects "Next to left bank station", the weir is assumed to be located on the left edge of the main channel. When the user selects "Next to right bank station", the weir is assumed to be located on the right edge of the main channel. When the user selects "Right overbank", the weir is assumed to be located at the right end of the cross section data.

Connected To. This field will display what the lateral structure is connected to (i.e. where the water leaving from the main river will be going). If the

structure is not connected to anything, this field will be blank. A lateral structure can be connected to a storage area, another river reach, or nothing at all. The buttons below this field are used to connect the lateral structure, as well as remove a connection. Whatever the lateral structure is connected to is where the water will be sent, and will also be used as the tailwater elevation for submerged hydraulic calculations. Water can also flow in the reverse direction through a lateral structure if the connected to location has a higher water surface than the from location.

Structure Type. This field is used to select the type of routing that will be used for this structure. There are two options, **Weir/Gates/Culverts/ Diversion Rating Curves** (the default) and **Linear Routing**. The default option is where the program calculated the flow across the structure by performing detailed hydraulic calculations for the weir, gated spillways, culverts, and any rating curve. The second option, Linear Routing, is a simplified method in which the user just puts in a linear routing coefficient. This coefficient can vary between 0.0 and 1.0, with 1.0 representing sending the maximum flow over the structure and 0.0 representing no flow. The linear routing method is a simple storage accounting method. This method can be very useful when the user has many lateral structures connected to storage areas, and a detailed flow calculation over each structure is not necessary. Also, the linear routing method is computationally faster and more stable. Typical values for the linear routing coefficient are from 0.05 to 0.2. However, this coefficient needs to be calibrated.

Culvert Flap Gates. The drop down box right above the Structure Type pertains to having flap gates on culverts. This option only affects the flow through the culverts, not the weir or the gated structures. The options are no flap gates (default), no negative flow flap, and no positive flow flap. No flap gates means that flow is allowed to go in both directions through the culvert. The no negative flow flap option means that flow can only go in the positive flow direction through the culverts. Positive flow is assumed to be taking flow away from the river for a lateral structure. Therefore, the no negative flow flap option would allow water to go away from the main river through the culverts, but not back into the river. The final option, no positive flow flap, would only allow water to come into the main river through the culverts, but not away from the main river.

Optimization. This option is for steady flow modeling only. When modeling in a steady flow mode, the user can have the software figure out how much flow will leave through the lateral structure, and how much will continue on downstream. This calculation requires an iterative solution. Pressing the Optimization button brings up an editor that allows the user to turn the optimization option on. When optimization is not turned on, the program will assume all of the water is still going downstream, though it will calculate what could have gone out the lateral weir based on the computed water surface. When optimization is turned on, the software calculates the flow out of the lateral structure, reduces the flow in the main river, and then recalculates the profile in the main river. This operation continues until there is a balance between the calculated and assumed flows for the main river.

Breach. This button allows the user to define information for evaluating the breaching of this lateral structure. The data is actually stored in the currently opened plan file. The editor can also be brought up from the plan editor. This option is only for unsteady flow modeling. To learn more about this option, see Chapter 16 "Advanced Features For Unsteady Flow Routing."

Weir/Embankment. The Embankment and Weir data are entered together, and are used to describe the embankment in which the outlets will be placed, as well as any uncontrolled weirs. To enter the weir and embankment data, press the **Weir/Embankment** button and the editor will appear as shown in Figure 6-32.

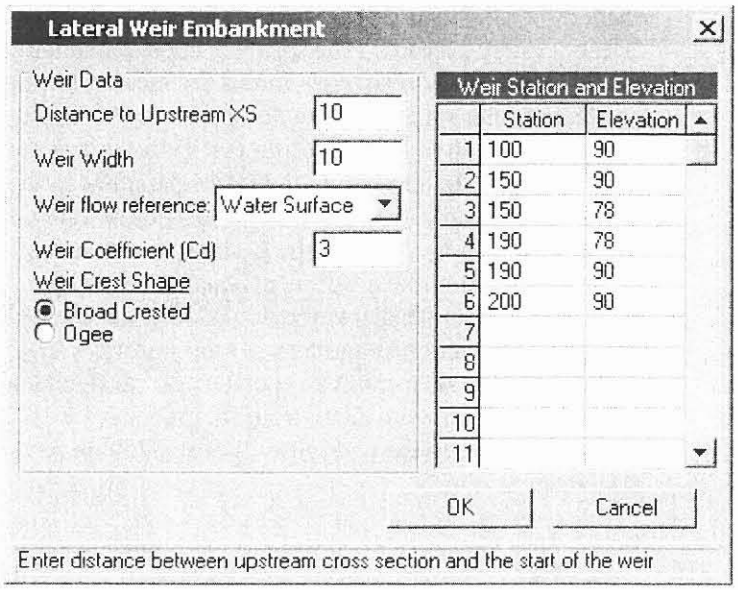


Figure 6.32 Lateral Weir/Embankment editor

The Lateral Weir/Embankment Data editor is similar to the Deck/Roadway editor for bridges and culverts. The data on the Weir/Embankment editor is the following:

Distance - The distance field is used to enter the distance between the upstream end of the Weir/Embankment (based on where the user will start to enter the embankment data) and the cross section immediately upstream of the structure. This distance is entered in feet (or meters for metric).

Weir Width - The width field is used to enter the width of the top of the embankment. This value will only be used for graphical plotting, and does not have any effect on the computations. The width of the embankment should be entered in feet (meters for metric).

Weir flow reference - This value is used to select whether weir flow is computed by using the energy gradeline or the water surface from the cross sections. The default is to use the energy gradeline.

Weir Coefficient - Coefficient that will be used for weir flow over the embankment in the standard weir equation.

Weir Crest Shape - When submergence occurs over the weir there are two choices available to figure out how much the weir coefficient should be reduced due to the submergence. These two criteria are based on the shape of the weir. The first method is based on work that was done on a trapezoidal shaped broad crested weir (FHWA, 1978). The second criterion was developed for an Ogee spillway shape (COE, 1965). The user should pick the criterion that best matches their problem. If the user selects the Ogee Spillway shape, then some additional information is required. For an Ogee shaped weir the user must enter the "Spillway Approach Height" and the "Design Energy Head". The spillway approach height is equal to the elevation of the spillway crest minus the mean elevation of the ground just upstream of the spillway. The design energy head is equal to the energy grade line elevation (at the design discharge) minus the elevation of the spillway crest. In addition to these two parameters, the user has the option to have the program calculate the weir coefficient at the design discharge. This is accomplished by pressing the $\mathbf{C_d}$ button. Once this button is pressed, the program will compute a weir coefficient for the Ogee spillway based on the design head. During the weir calculations, this coefficient will fluctuate based on the actual head going over the spillway. The curves used for calculating the Ogee spillway coefficient at design head, and discharges other than design head, were taken from the Bureau of Reclamation publication "Design of Small Dams", Figures 249 and 250 on page 378 (Bureau of Reclamation, 1977).

Station and Elevation Coordinates - This table is used to define the geometry of the Weir and the Embankment. The information is entered from upstream to downstream in stationing. The user enters stations and elevations of the top of the embankment and weir. The stationing does not have to equal the stations in the river reach. Everything below these elevations will be filled in to the ground. The **Del Row** and **Ins Row** buttons allow the user to delete and insert rows.

Gated Spillway Data. In addition to uncontrolled overflow weirs, the user can add gated spillways (this is optional). To add gated spillways to the structure, press the **Gate** button on the Lateral Weir and Gated Spillway data editor. Once this button is pressed, the lateral gated editor will appear as shown in Figure 6.33 (except yours will be blank until you have entered some data).

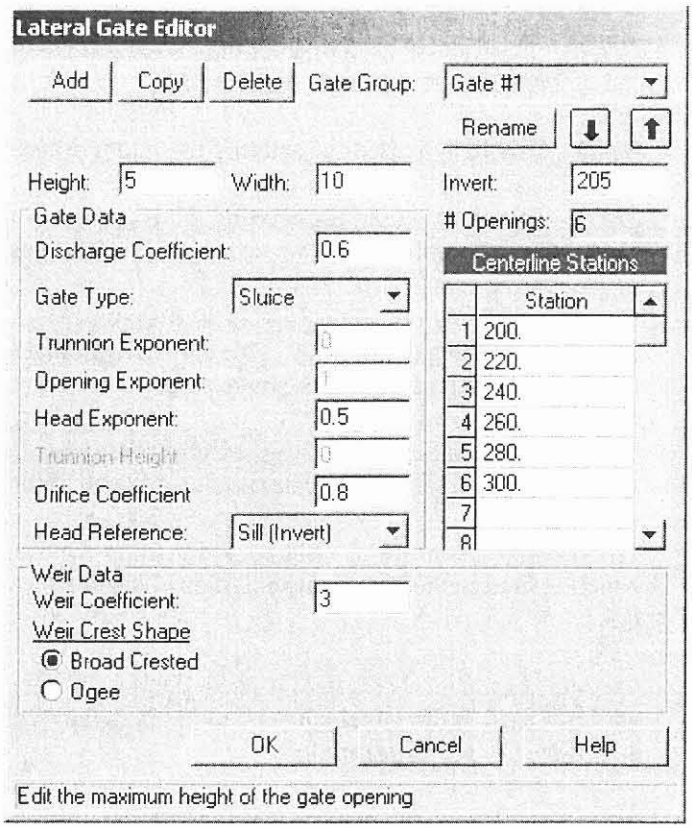


Figure 6.33 Lateral Gated Spillway Editor.

The Gated Spillway editor is similar to the Culvert editor in concept. The user enters the physical description of the gates, as well as the required coefficients, in the Gated Spillway editor. The functionality of the gates is defined as part of the Steady Flow data, on a per profile basis. The following is a list of the data contained on this editor:

Gate Group - The Gate Group is automatically assigned to "Gate #1" the first time you open the editor. The user can enter up to 10 different Gate Groups at each particular structure, and each gate group can have up to 25 identical gate openings. If all of the gate openings are exactly the same, then only one gate group needs to be entered. If the user has gate openings that are different in shape, size, elevation, or have different coefficients, then additional Gate Groups must be added for each Gate type. To add an additional gate group, you can either use the **Add** or **Copy** buttons. The **Add** button increments the Gate # and clears the gate editor. The **Copy** button increments the Gate # and makes a copy of the original Gate group data. Once a copy is made of the gate data, the user can change any of the existing gate information. Gate groups can be deleted by pressing the **Delete** button. Also, if the gates are identical, but the user wants to be able to open the gates to different elevations, then the user must have a separate gate group for each set of gates that will be opened to different elevations.

Height - This field is used to enter the maximum possible height that the gate can be opened in feet (meters for metric).

Width - This field is used for entering the width of the gate in feet (meters).

Invert - This field is used for entering the elevation of the gate invert (sill elevation of the spillway inside of the gate) in feet (meters for metric).

Discharge Coefficient - This field is used for entering the coefficient of discharge for the gate opening. This coefficient ranges from 0.6 to 0.8 for Radial gates and 0.5 to 0.7 for sluice gates.

Gate Type - This field is used for selecting the type of gate. Two gate types are available, radial (tainter gate) or sluice (vertical lift gate).

Trunnion Exponent - This field is used to enter the trunnion height exponent, which is used in the radial gate equation. The default value for this field is 0.0.

Opening Exponent - This field is used to enter the gate opening exponent, which is used in the radial gate equation. A default value of 1.0 is automatically set for this field.

Head Exponent - This field is used to enter the upstream energy head exponent, which is used in the radial gate equation. A default value of 0.5 is automatically set for this field.

Trunnion Height - This field is used for entering the height from the spillway crest to the trunnion pivot point. See Chapter 8 of the Hydraulic Reference manual for more details on this variable.

Orifice Coefficient - This field is used to enter an orifice coefficient, which will be used for the gate opening when the gate becomes more than 80 percent submerged. Between 67 percent and 80 percent submerged, the program uses a transition between the fully submerged orifice equation and the free flow equations. When the flow is less than 67 percent submerged, the program uses the free flow gate equations.

Head Reference – This field is used to select the reference point for which the upstream energy head will be computed from. The default is the gate sill (invert), which is normally used when the flow through the gate goes out into a channel. If the gate causes the flow to jet out freely into the atmosphere, then the head reference should be selected as the centerline elevation of the gate opening. If the gate crest is an ogee spillway crest, then the center of the gate opening should be used. Ogee spillway crests are normally designed to follow the shape of water jetting freely into the atmosphere.

Centerline Stations - This table is used for entering the centerline stationing of the identical gate openings. The user should enter a different centerline stationing for each gate opening that is part of the current gate group. All

gate openings within the same gate group are exactly identical in every way, except their centerline stationing. As a user adds new centerline stationing values, the number of identical gates in the group is automatically incremented and displayed in the field labeled "# Openings".

Weir Coefficient - This field is used for entering a weir coefficient that will be used for the gate opening. This coefficient will only be used when the gate is opened to an elevation higher than the upstream water surface elevation. When this occurs, the flow through the gate is calculated as weir flow.

Weir Crest Shape - This parameter allows the user to select between a Broad Crested shape weir and an Ogee shaped weir. Depending on which shape is selected, the program will use a different submergence criteria during the calculation. In addition to the submergence criteria, if the user selects the Ogee shape, the program will bring up two additional data entry fields that must be entered by the user. These fields are used for the Spillway Approach Height and the Design Energy Head, which are explained below. Once these fields are entered, the user should press the button labeled $\mathbf{C_d}$. When this button is pressed, the program will compute a weir coefficient for the Ogee spillway based on the design head. During the weir calculations, this coefficient will fluctuate based on the actual head over the gated spillway. The curves used for calculating the Ogee spillway coefficient at design head, and discharges other than design head, were taken from the Bureau of Reclamation publication "Design of Small Dams", Figures 249 and 250 on page 378 (Bureau of Reclamation, 1977).

Spillway Approach Height - The spillway approach height is equal to the elevation of the spillway crest minus the mean elevation of the ground just upstream of the spillway.

Design Energy Head - The design energy head is equal to the energy grade line elevation (at the design discharge) minus the elevation of the spillway.

Once all of the data for the gates have been entered, the user needs to press the **OK** button for the data to be accepted. If the user does not want to use the new data, and would like to go back to the original data they had before entering the Gate Editor, press the **Cancel** button. If the user presses the OK button, this does not mean that the data is saved to the hard disk, it is only stored in memory and accepted as being good data. This data is part of the geometry data, and is stored in the geometric data file. The data can be stored to the hard disk by selecting one of the save options from the File menu of the Geometric Data window.

Culvert Data. In addition to the lateral weir and gates, the user can also enter lateral culverts. To add culverts to the structure, press the **Culvert** button on the lower left side of the editor. When this option is selected the following window will appear.

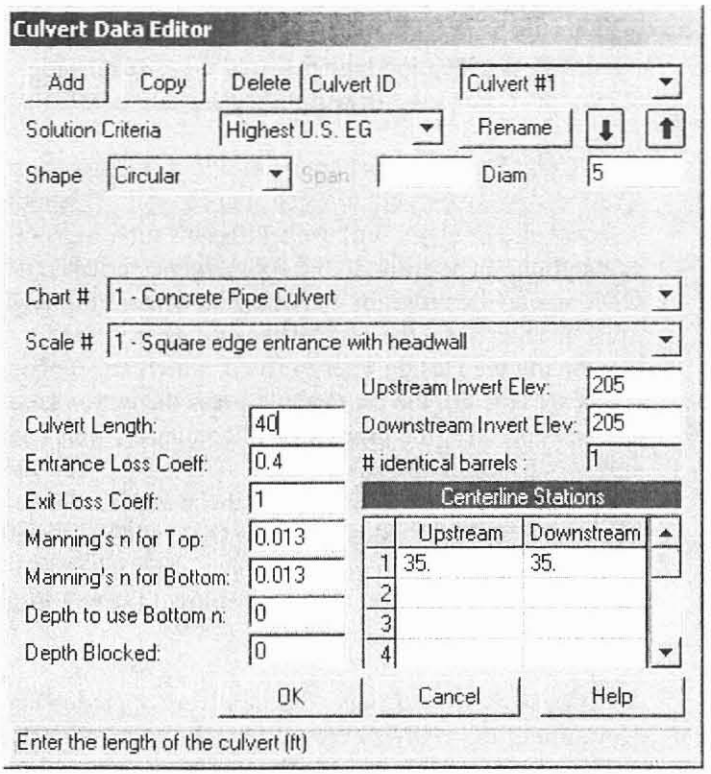


Figure 6.34 Culvert Editor for Lateral Culverts.

The required culvert data is the same as for an inline culvert. To see an explanation of each field on the editor, review the information on culverts found earlier in this chapter. The only difference is that the centerline stationing of each culvert is based on the stationing entered in the Weir/Embankment editor.

Diversion Rating Curve. Diversion rating curves are used to remove flow from a main river. The diversion, rating curve can be used in conjunction with a lateral weir, gated structures, and culverts, or it can be used alone.

To add a diversion, rating curve to the system, press the "**Diversion Rating Curve**" button on the left hand side of the Lateral Structure editor. When this button is pressed, the following editor will appear:

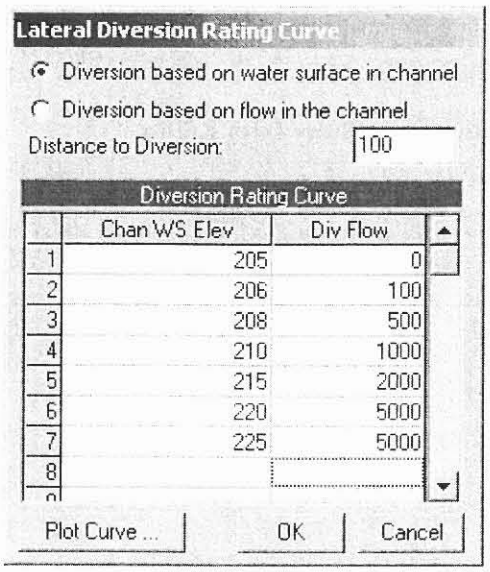


Figure 6.35 Lateral Rating Curve Editor

The user first selects the type of rating to be used: channel water surface versus diverted flow or channel flow versus diverted flow. Next, the distance between the location of the diversion and the cross section just upstream of the structure must be entered in order to locate the diversion. Finally, the user enters the actual rating curve. The curve is entered as the amount of flow leaving, verses the elevation of the water in the main river or flow in the main river.

Linear Routing Option

The user can choose to use a linear routing option instead of entering structure information and having the program compute the flow from the structures. The linear routing option is selected by going to the Structure Type pull down and selecting "**Linear Routing**" from the list. This option uses a coefficient times the difference in available storage between the too and from connection. When this option is selected, a linear routing button will appear on the left side of the window. Selecting the linear routing button will bring up the following window:

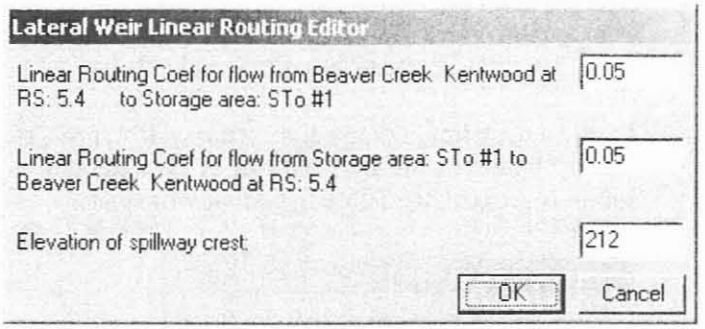


Figure 6.43 Simple Spillway Data Editor

As shown in the Linear Routing editor, the user must enter a linear routing coefficient for both the positive and negative flow directions. Additionally, the minimum elevation of the spillway crest must be entered. If both water surfaces go below the spillway crest, no flow is passed over the structure.

Storage Areas

Storage Areas are lake like regions in which water can be diverted into or from. Storage areas can be located at the beginning of a reach, at the end of a reach, or they can be located laterally to a reach. Storage areas can be connected to a river reach by using a lateral structure connection. Storage areas can be connected to another storage area by using a storage area connection. Storage area connections consist of a weir, gated spillways and a weir, culverts and a weir, or a linear routing option. An example river system with storage areas is shown in Figure 6.36.

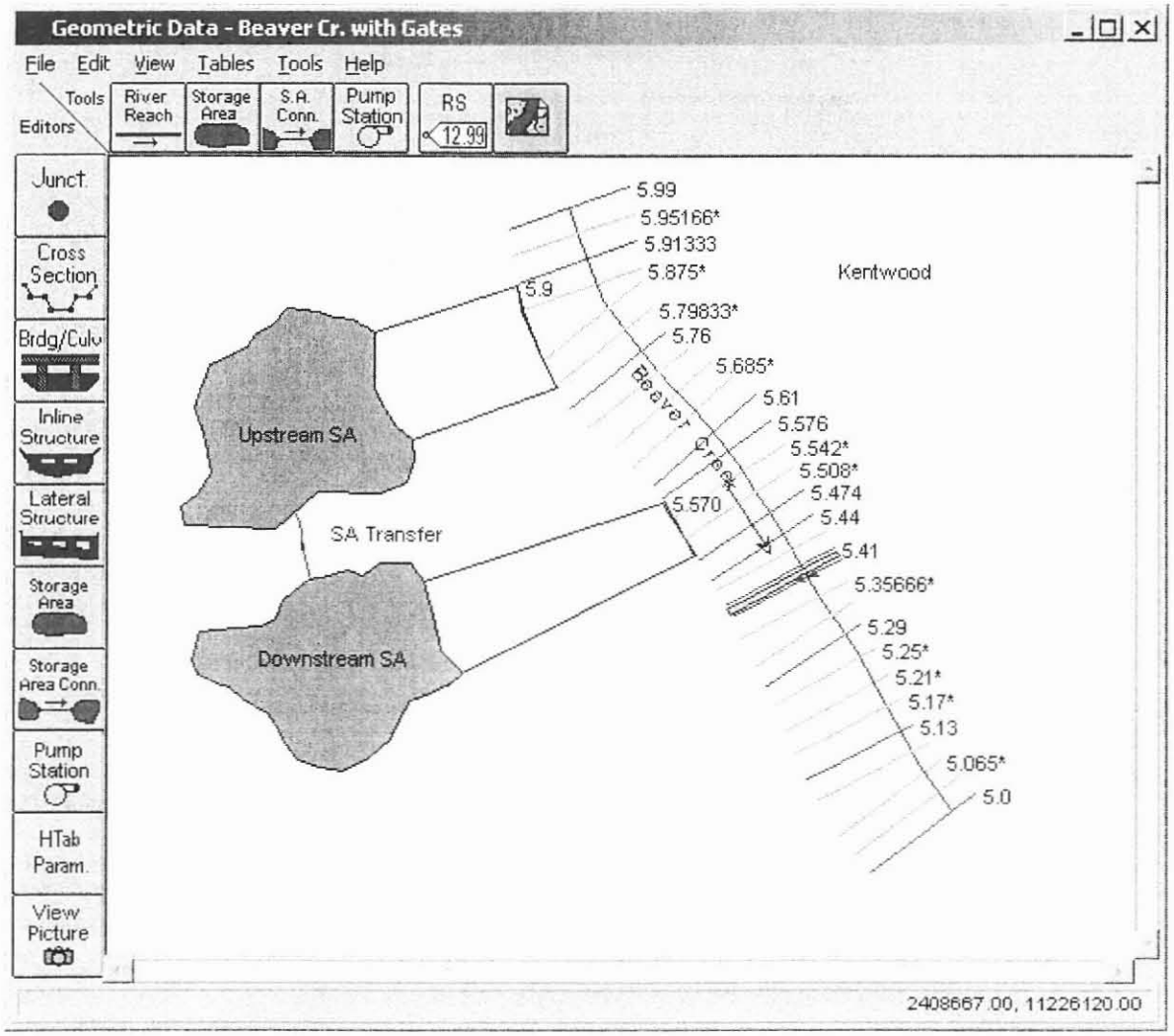


Figure 6.36 Example Schematic With Storage Areas.

To add a storage area to the river system, first use the storage area drawing tool at the top of the geometric editor. Once the storage area, drawing tool is

selected, the user single clicks the left mouse button to begin drawing the storage area. Additional points are added by moving the mouse and single clicking. The storage area will be represented as a polygon. To finish drawing the storage area, double click the left mouse button. The first and last point will then be connected, and the storage area will be filled in with a blue color.

After the storage area is drawn, the user must enter data to describe the storage area. This is accomplished with the storage area editor, which is one of the buttons on the left side of the geometric editor. Press the storage area editor button and the following editor will appear:

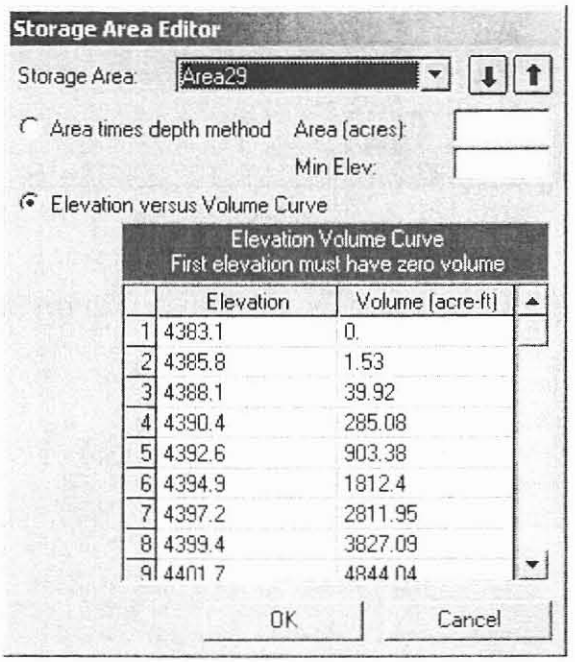


Figure 6.37 Storage Area Editor

As shown in the figure above, the user has two options for entering information about the volume of the storage area. The first option is a simple area times a depth option. The user enters the area of the storage, and a minimum elevation. The storage area is assumed to have the same area at all elevations, therefore the volume is dimply the depth times the area. The second option is to enter and elevation versus volume relationship for the storage area. This option provides more detail and accuracy, and is the recommended method whenever possible.

Storage Area Connections

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

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

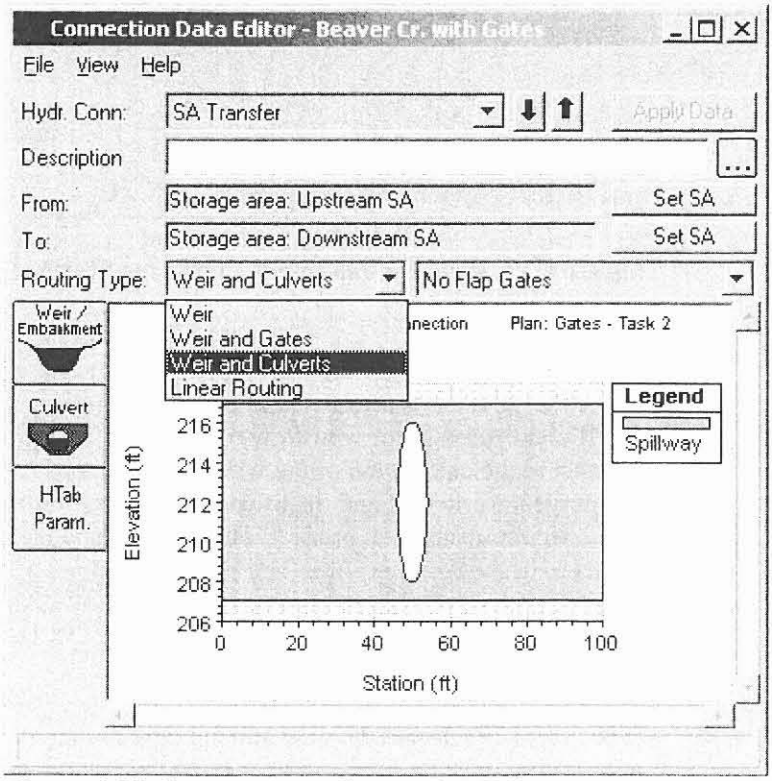


Figure 6.38 Storage Area Connection Editor

As shown in the figure above, the user should first enter a description for the storage area connection. Next the type of routing connection must be selected from the drop down box. As mentioned previously, the user has the choice of a weir only; weir and gates; weir and culverts; or linear coefficient routing between storage areas. Once a routing type is selected, the window will place

editor buttons specific to that type of routing onto the left side of the editor. In this example, because a weir and culverts was selected, the weir, culverts, and hydraulic table parameters buttons show up on the window. The user must enter data for each of the buttons shown on the window. In this example it is required to enter a weir/embankment, culvert information, and hydraulic table parameters. When the weir/embankment editor is selected, the following window will appear:

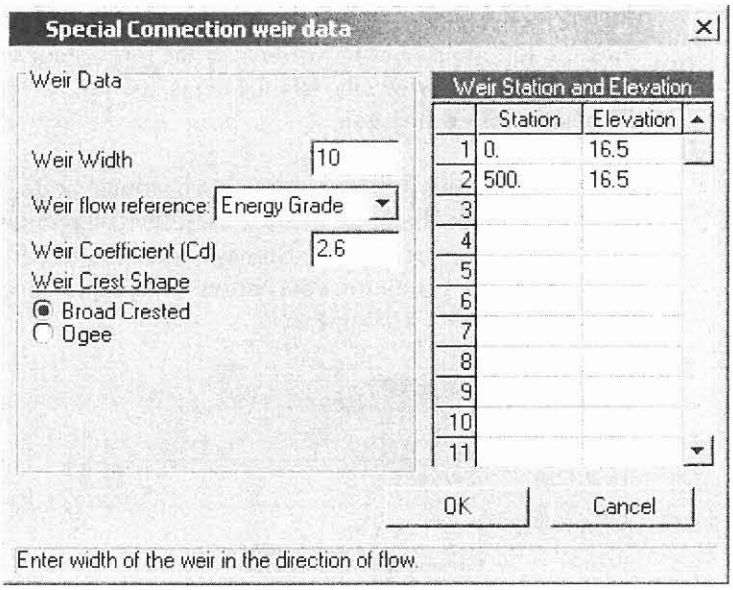


Figure 6.39 Weir/Embankment Editor for Storage Area Connections

To complete the data for the weir/embankment between two storage areas, the user enters a **Weir Width** (used only for drawing the schematic); a **Weir Coefficient** (used in the weir flow calculations); a **Weir Crest Shape** (used to assist in the calculation of the weir coefficient, as well as defining submergence criteria); and the **Station/Elevation Points** that describe the top of the weir/embankment profile. The weir/embankment can have up to 500 points to describe the profile. The program will use all of the information entered by the user for calculating weir flow between the two storage areas. After all of the data is entered, simply press the **OK** button to have the data accepted by the program.

If the user has selected the weir and culverts routing type, then culvert information must be entered also. When the culvert editor button is selected, the following window will appear:

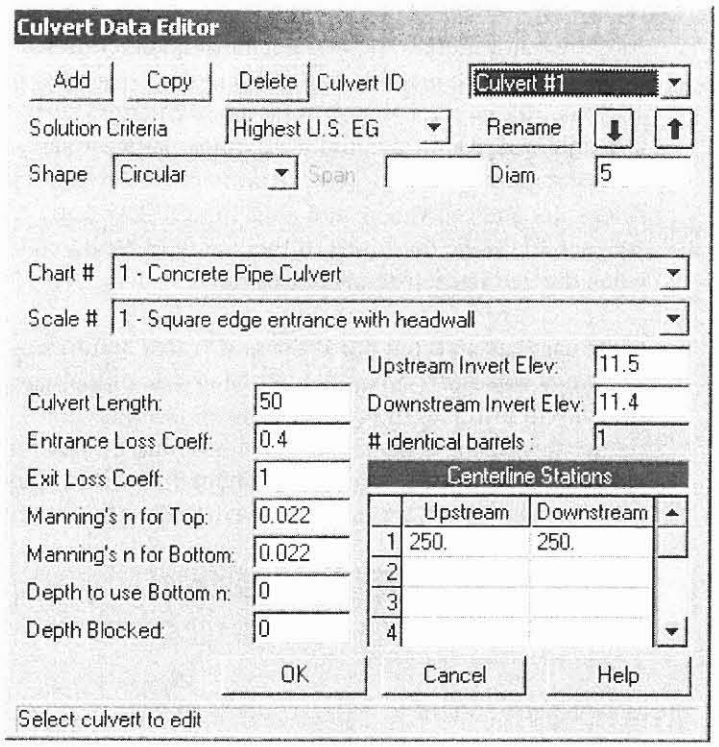


Figure 6.40 Culvert Editor for Storage Area Connections

The culvert editor shown above has exactly the same information as the normal culvert editor used at a river crossing. For detailed information about this editor, please review the section on culverts found earlier in this chapter.

Whenever the user selects a weir, or a weir and culvert routing type, and additional button will appear on the editor labeled "**Htab Param**". This editor is used to define the limits of the family of rating curves that will be developed for the storage area connection. When this button is pressed, the following editor will appear:

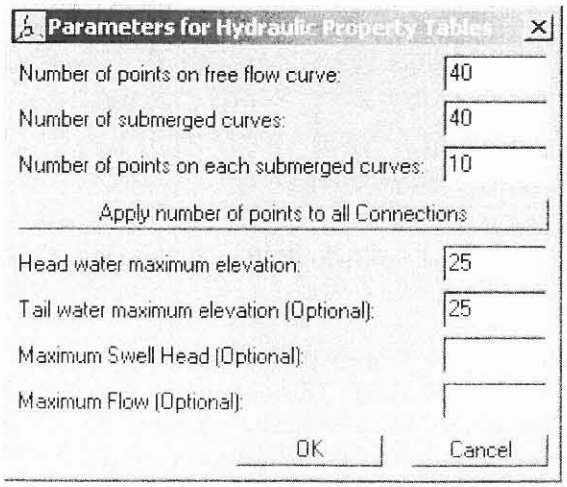


Figure 6.41 Hydraulic Properties Table Editor

As shown in Figure 6.41, the user must enter a number of points for the free flow curve (default is 40, max 50); number of submerged curves (default is 40, max 50); number of points on the submerged curves (default is 10, max 20); and head water maximum elevation. Additionally, the user can enter a tailwater maximum elevation; maximum swell head (difference between headwater and tailwater); and a maximum flow rate. All of these parameters are used to define the limits of the family of rating curves that get created when the geometric pre-processor runs.

If the user has selected the **Weir and Gates** option, an editor for gated spillways will show up on the left hand side of the window. Pressing this button will bring up the window shown in Figure 6.42. As shown in the figure, this editor is the same gate editor that is used for inline and lateral gated spillways. For information about this editor, please review the sections on inline and lateral gated spillways found earlier in this chapter.

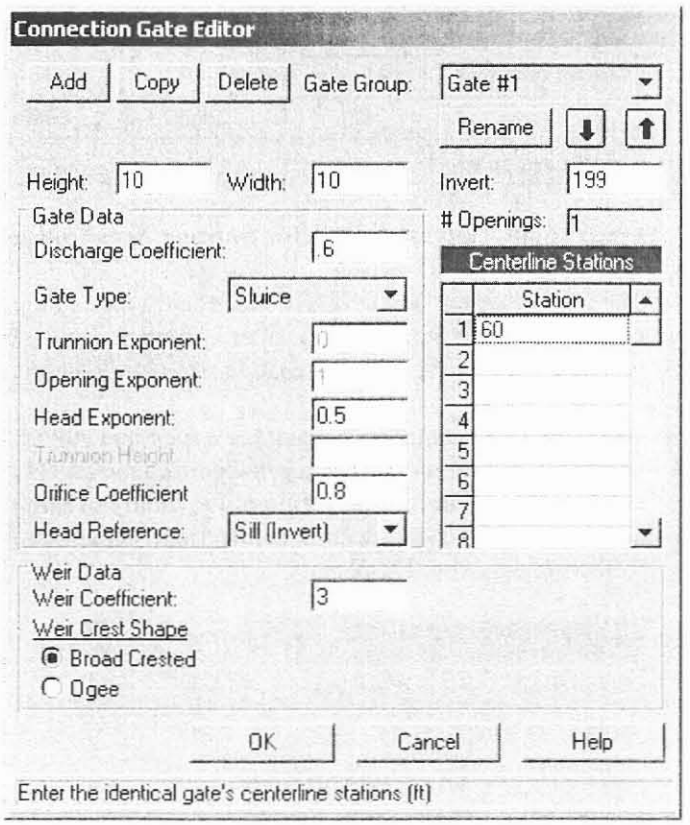


Figure 6.42 Gated Spillway Editor for Storage Area Connections

The final option for Storage Area Connections is to model the connection as a **Linear Routing Method**. This option uses a coefficient times the difference in available storage between the two storage areas, divided by the time step. When this option is pressed, the following window will appear:

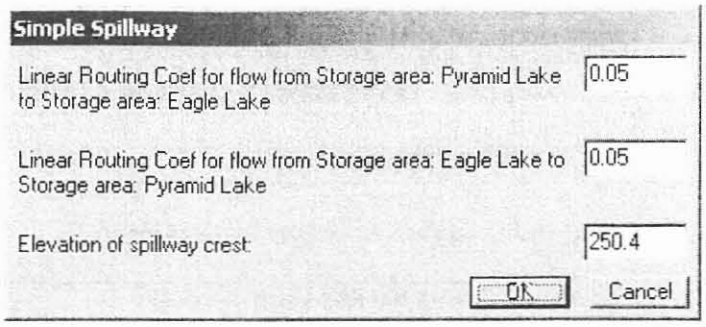


Figure 6.43 Simple Spillway Data Editor

As shown in the Linear Routing editor, the user must enter a linear routing coefficient for both the positive and negative flow directions. Additionally, the minimum elevation of the spillway crest must be entered. If both water surfaces go below the spillway crest, no flow is passed between the storage areas.

Pump Stations

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

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

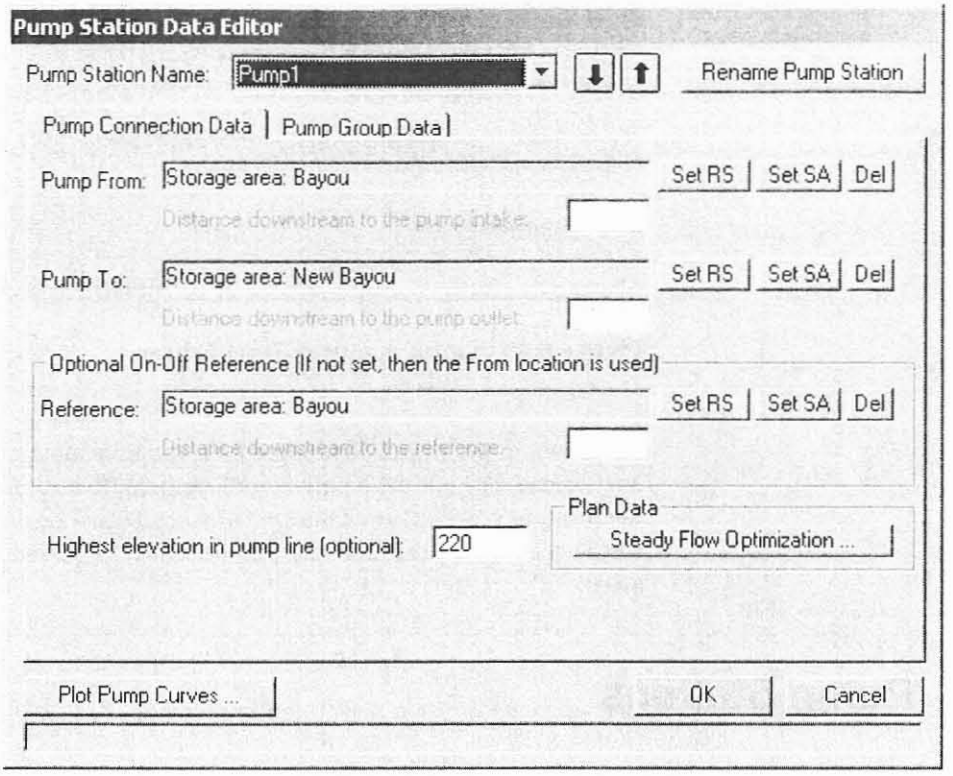


Figure 6.44 Pump Station Editor with Pump Connection Data

As shown in Figure 6.44, there are two tabs on the pump station editor, one is for the pump connection data, and the other is for the pump group data. The pump connection data contains the following data:

Rename Pump Station: This option allows the user to rename the pump station to something other than the default.

Pump From: This is the location of where the pump station is pumping from. This can be either a storage area or a river station from a river reach.

Pump To: This is the location of where the pump station is pumping to. This can be either a storage area or a river station from a river reach.

On-Off Reference: By default the program uses the "Pump From" location to determine when the pump should turn on or off. However, the user has the option to set a different location to be used as the monitor point for determining whether the pump should be turned on or off. This optional monitor location can be a storage area or a river station within a river reach.

Highest elevation in pump line: This option allows the user to enter an elevation to be used as the highest elevation in the pump line. One example of where this may be useful is if a pump station was being used to pump water over top of a levee. In this situation, the to and from water surface elevations does not completely quantify the required head to pump the water

over the levee. So it is necessary to enter the elevation of the highest point in the pump line (top of the levee) in order to accurately compute the flow going through the pump.

Steady Flow Optimization: This option is for steady flow modeling only. If water is being pumped from or to a river reach, the amount of flow going into or out of the reach should be accounted for when computing the water surface profiles. However, the water surface profiles will affect the computation of the amount of flow through the pumps. Therefore, to calculate this accurately, the pump flow and water surface profiles must be calculated iteratively until a balance is found between the river flows and the pump flows. This optimization feature is not done automatically by the steady flow program, however, the user can have the program do this by selecting **Steady flow optimization.** When this option is selected, a window will appear allowing the user to turn the pump flow optimization on.

In addition to the pump connection data the user must fill out the pump group data. Select the **Pump Group Data** tab and the editor will look like the following:

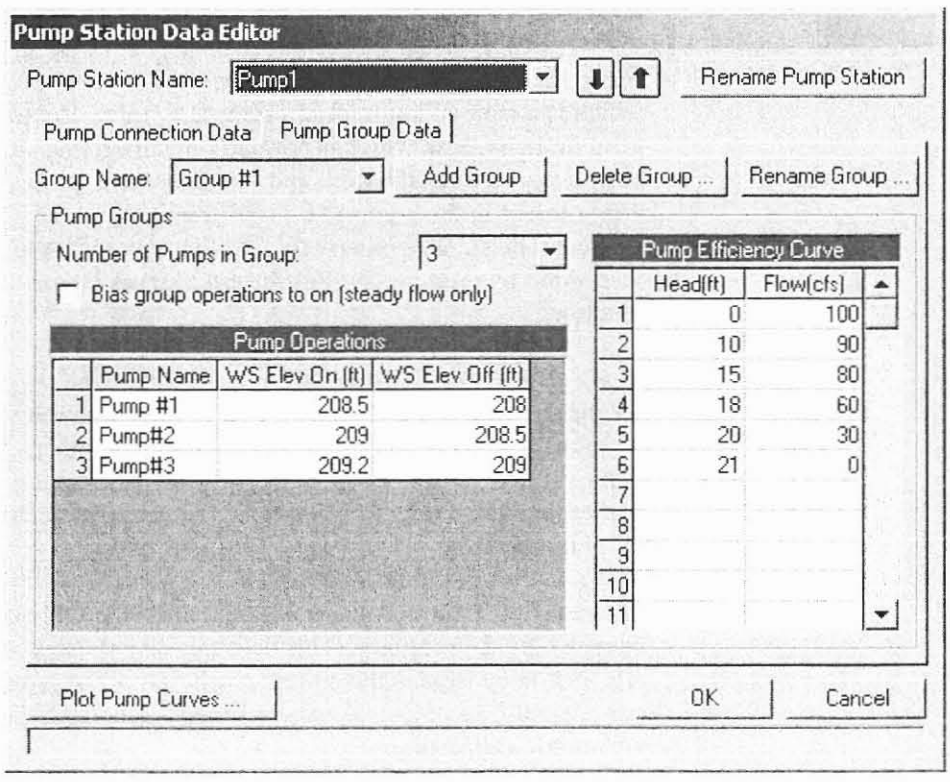


Figure 6.45 Pump Station Editor With Pump Group Data.

As shown in Figure 6.45, the pump group data consists of the following:

Group Name: By default the first pump group is called "Group #1", and the second would be "Pump Group #2", etc. The user has the option to rename

any pump group to whatever they would like. This is done by pressing the "Rename Group" button.

Add Group: This button is used to add another pump group. If you have pumps that have different flow capacities and use different pump efficiency curves, they must be entered as a separate pump group.

Delete Group: This button is used to delete the current pump group.

Number of Pumps in Group: This field is used to enter the number of identical pumps in the current pump group. Identical pumps must use the same pump efficiency curve but can have different on and off trigger elevations.

Bias group operations to on (Steady Flow Only): This option is only relevant for a steady flow run. When this option is selected, and a particular water surface profile is between the on and off elevation for a pump, the program will assume the pump is turned on. If this option is not checked, then the program will assume the pump is off when the water surface is between the on and off elevations.

Pump Efficiency Curve: This table is used to enter the pump efficiency curve, which is a table of pump head versus flow rate. The head represents the total head in the system, which is normally the difference in the water surface elevations between the from and the to location, plus any friction losses and minor losses in the pump line. The flow is the pump rate capacity at that particular head. The pump efficiency curve can be plotted for visual inspection by pressing the **Plot Pump Curves** button at the bottom of the window.

Pump Operations: This table is used to define the trigger elevations for when the pumps will turn on and off. The monitor location for triggering a pump on or off is by default the from location, unless otherwise specified in the Optional On-Off Reference field. In general, the pump on elevation must be higher than the pump off elevation. Trigger elevations must be specified for all of the pumps.

After all of the pump data are entered, press the **OK** button to have the data excepted by the program. This does not save the data to the hard disk, it only allows it to be used in the current execution of the program. To save the data permanently, you must save the geometry data from the File menu of the Geometric Data Editor.

Cross Section Interpolation

Occasionally it is necessary to supplement surveyed cross section data by interpolating cross sections in between two surveyed sections. Interpolated cross sections are often required when the change in velocity head is too large to accurately determine the energy gradient. An adequate depiction of the change in energy gradient is necessary to accurately model friction losses as well as contraction and expansion losses.

Cross section interpolation can be accomplished in three ways from within the HEC-RAS interface. The first method is to simply copy one of the bounding cross sections and then adjust the station and elevation data. The cross section editor allows the user to raise or lower elevations and to shrink or expand various portions of any cross section. The second and third options allow for automatic interpolation of cross section data. From the Geometric Data editor, automatic interpolation options are found under the **Tools** menu bar as shown in Figure 6.46.

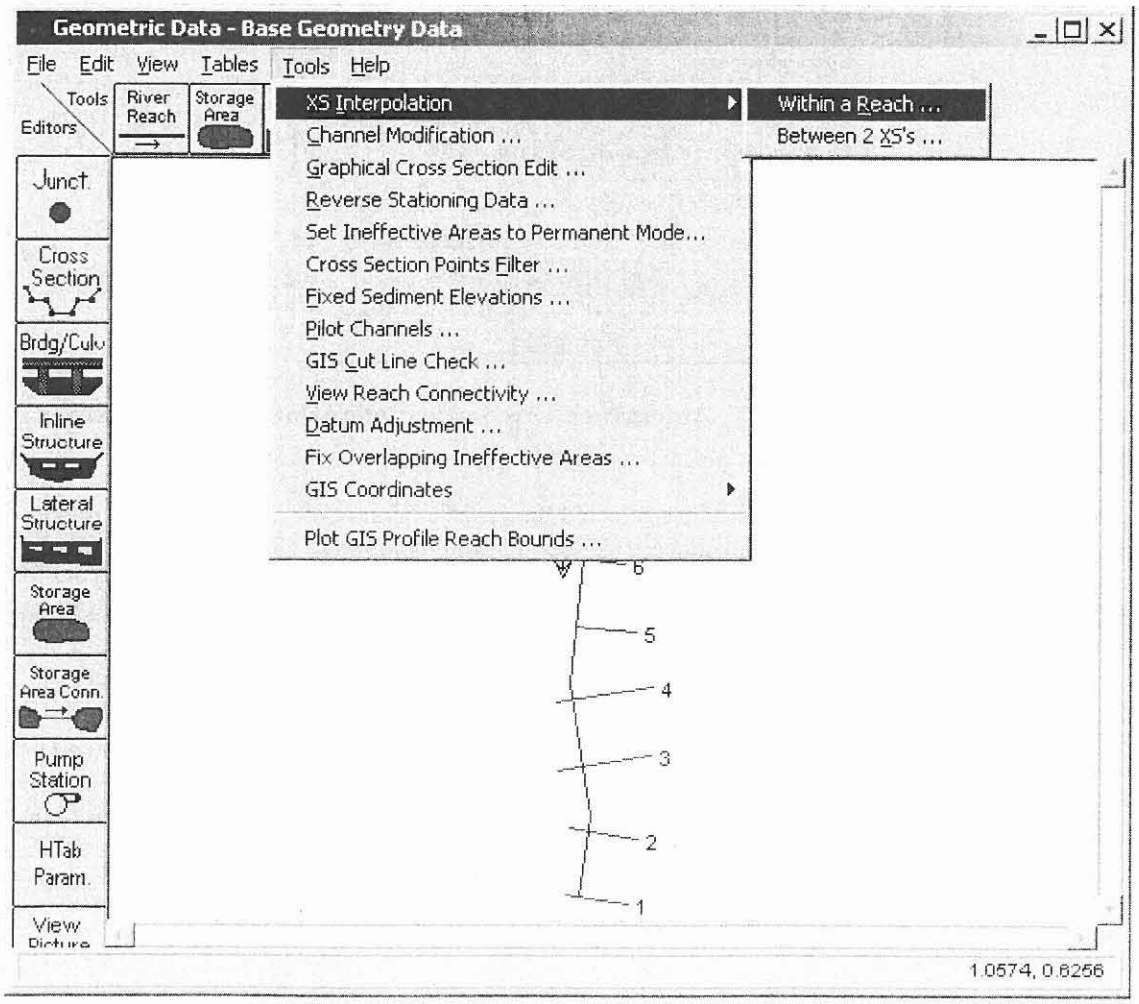


Figure 6.46 Automatic Cross Section Interpolation Options

The first cross section interpolation option, **Within a Reach**, allows for automatic interpolation over a specified range of cross sections within a single reach. When this option is selected, a window will pop up as shown in Figure 6.47. The user must first select the River and Reach that they would like to perform the interpolation in. Next the user must select a starting River Station and an ending River Station for which interpolation will be performed. The user must also provide the maximum allowable distance between cross sections. If the main channel distance between two sections is greater than the user defined maximum allowable, then the program will interpolate cross sections between these two sections. The program will interpolate as many cross sections as necessary in order to get the distance between cross sections below the maximum allowable. Additionally the user can specify the number of decimal places used for the stationing and elevations of the interpolated cross sections.

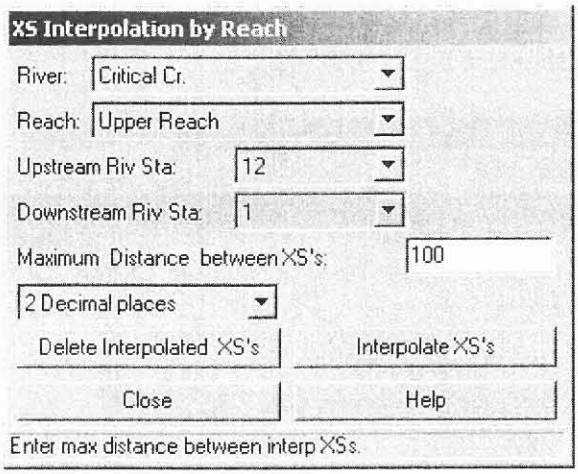


Figure 6.47 Automatic Cross Section Interpolating Within a Reach

Once the user has selected the cross section range and entered the maximum allowable distance, cross section interpolation is performed by pressing the **Interpolate XS's** button. When the program has finished interpolating the cross sections, the user can close the window by pressing the **Close** button. Once this window is closed, the interpolated cross sections will show up on the river schematic as light green tic marks. The lighter color is used to distinguish interpolated cross sections from user-entered data. Interpolated cross sections can be plotted and edited like any other cross section. The only difference between interpolated sections and user-defined sections is that interpolated sections will have an asterisk (*) attached to the end of their river station identifier. This asterisk will show up on all input and output forms, enabling the user to easily recognize which cross sections are interpolated and which are user defined.

The second type of automatic cross section interpolation, **Between 2 XS's**, allows the user to have much greater control over how the interpolation is performed. When this option is selected, a Cross Section Interpolation window will appear as shown in Figure 6.48.

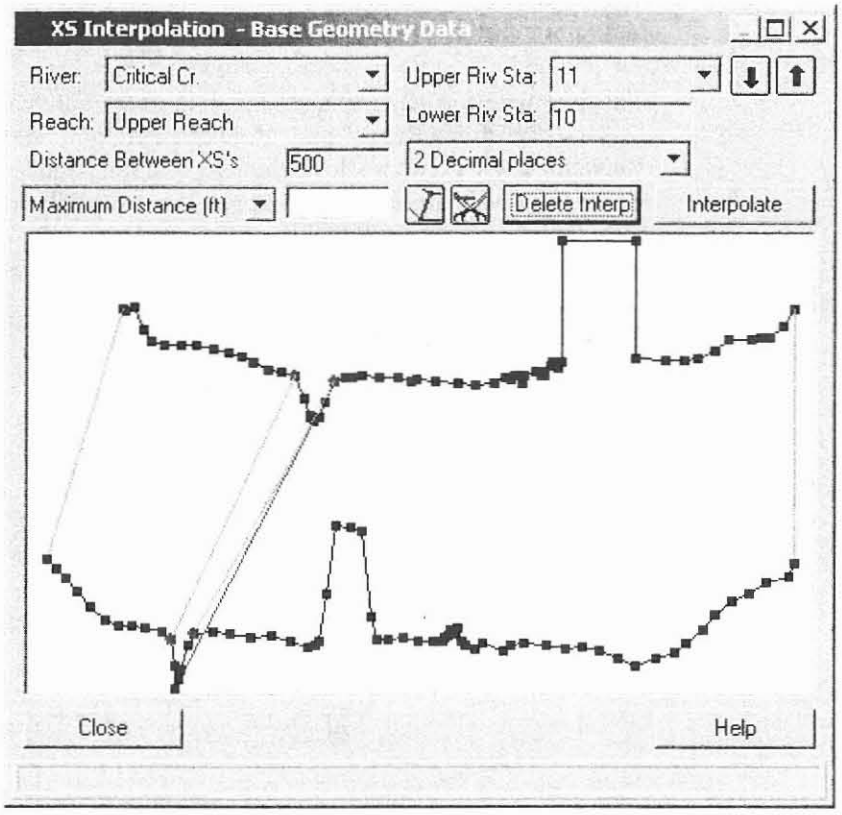


Figure 6.48 Detailed Cross Section Interpolation Window

This cross section interpolation window displays only two cross sections at a time. The user can get to any two cross sections from the River, Reach and River Station boxes at the top of the window. Interpolated cross section geometry is based on a string model as graphically depicted in Figure 6.46. The string model consists of chords that connect the coordinates of the upstream and downstream cross sections. The cords are classified as master and minor cords. As shown in Figure 6.48, five master cords are automatically attached between the two cross sections. These master cords are attached at the ends of the cross sections, the main channel bank stations, and the main channel inverts. Minor cords are generated automatically by the interpolation routines. A minor cord is generated 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 value at the opposite cross section is determined by computing the decimal percent that the known coordinate represents of the distance between master cords and then applying that percentage to the opposite cross section master cords. The number of

minor cords will be equal to the sum of all the coordinates of the upstream and downstream sections minus the number of master cords. Interpolation at any point in between the two sections is then based on linear interpolation of the elevations at the ends of the master and minor cords. Interpolated cross sections will have station and elevation points equal to the number of major and minor cords.

This interpolation scheme is used in both of the automated interpolation options ("Within a Reach" and "Between 2 XS's"). The difference is that the **Between 2 XS's** option allows the user to define additional master cords. This can provide for a better interpolation, especially when the default of five major cords produces an inadequate interpolation. An example of an inadequate interpolation when using the default cords is shown below.

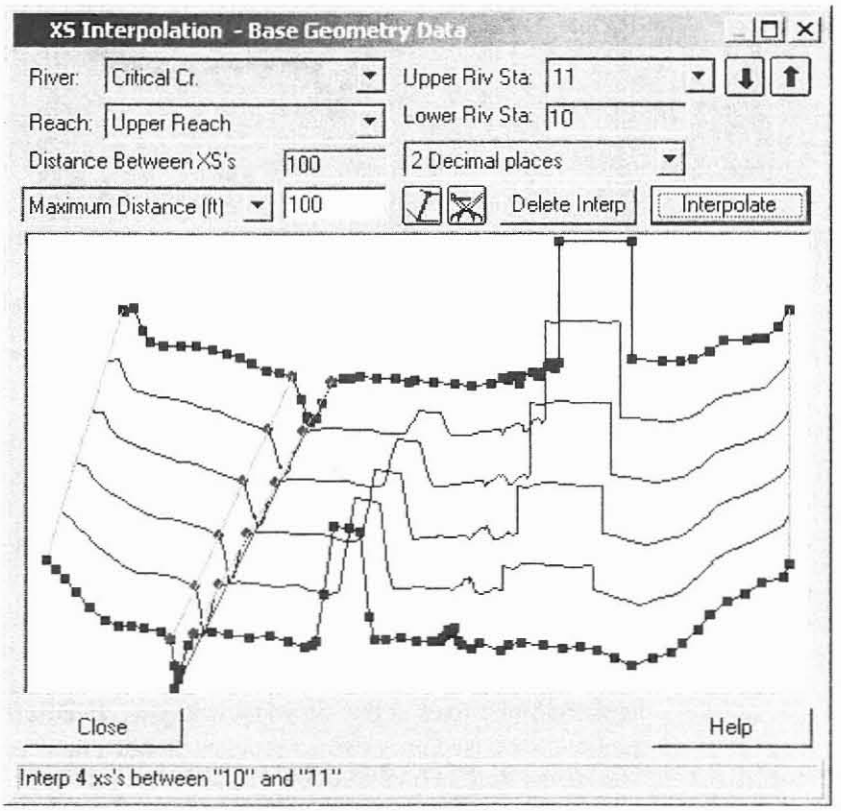


Figure 6.49 Cross Section Interpolation Based on Default Master Cords

As shown in Figure 6.49, the interpolation was adequate for the main channel and the left overbank area. The interpolation in the right overbank area failed to connect two geometric features that could be representing a levee or some other type of high ground. If it is known that these two areas of high ground should be connected, then the interpolation between these two sections should be deleted, and additional master cords can be added to connect the two features. To delete the interpolated sections, press the **Del Interp** button.

Master cords are added by pressing the **Master Cord** button that is located to the right of the Maximum Distance field above the graphic. Once this button is pressed, any number of master cords can be drawn in. Master cords are drawn by placing the mouse pointer over the desired location (on the upper cross section), then while holding the left mouse button down, drag the mouse pointer to the desired location of the lower cross section. When the left mouse button is released, a cord is automatically attached to the closest point near the pointer. An example of how to connect master cords is shown in Figure 6.50.

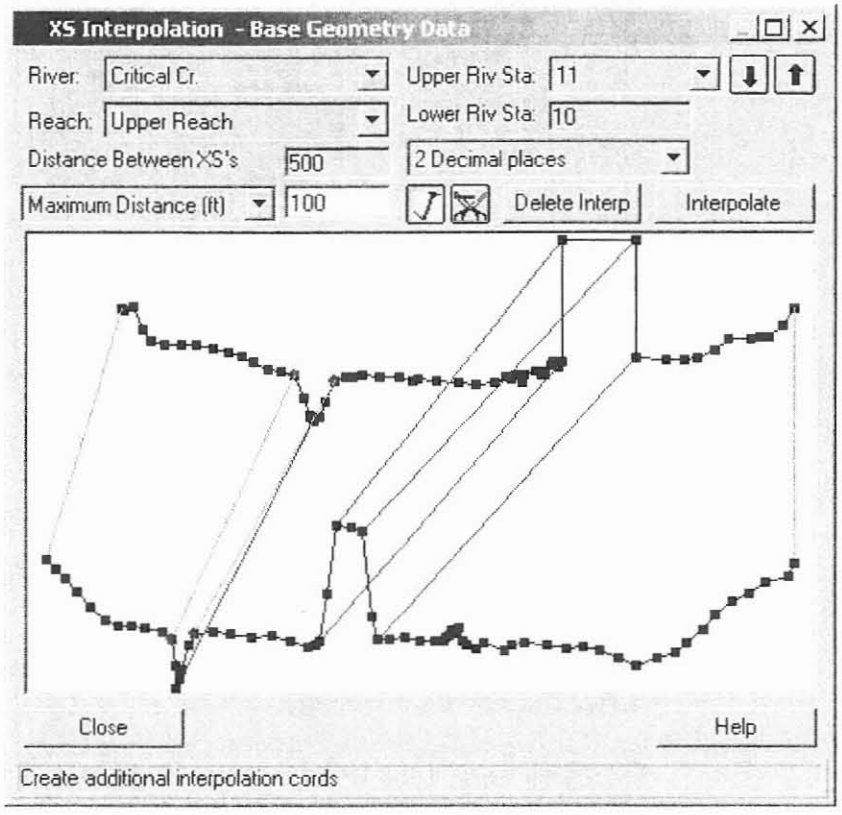


Figure 6.50 Adding Additional Master Cords for Interpolation

User defined master cords can also be deleted. To delete user defined master cords, press the **scissors** button to the right of the master cords button. When this button is pressed, simply move the mouse pointer over a user defined cord and click the left mouse button to delete the cord.

Once you have drawn in all the master cords that you feel are required, and entered the maximum distance desired between sections, press the **interpolate** button. When the interpolation has finished, the interpolated cross sections will automatically be drawn onto the graphic for visual inspection. An example of this is shown in Figure 6.51.

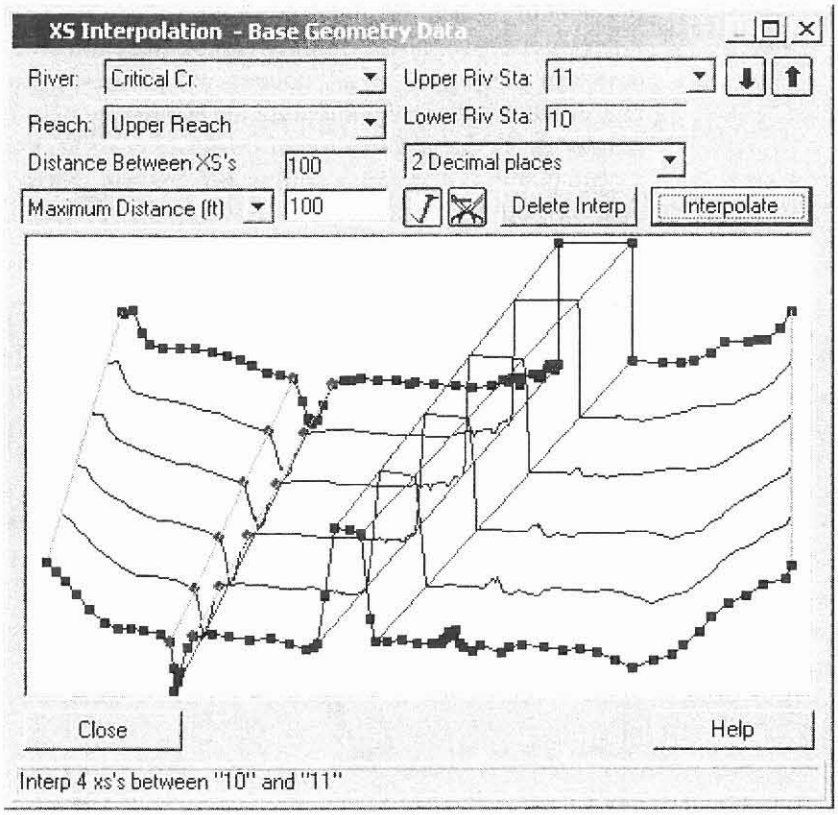


Figure 6.51 Final Interpolation With Additional Master Cords

As shown in Figure 6.51, the interpolation with the addition of user defined master cords is very reasonable.

In general, the best approach for cross section interpolation is to first interpolate sections using the "**Within a Reach**" method. This provides for fast interpolation at all locations within a reach. The "Within a Reach" method uses the five default master cords, and is usually very reasonable for most cross sections. Once this is accomplished, all of the interpolated sections should be viewed to ensure that a reasonable interpolation was accomplished in between each of the cross sections. This can be done from the **"Between 2 XS's"** window. Whenever the user finds interpolated cross sections that are not adequate, they should be deleted. A new set of interpolated cross sections can be developed by adding additional master cords. This will improve the interpolation.

CAUTION: Automatic geometric cross section interpolation should not be used as a replacement for required cross section data. If water surface profile information is required at a specific location, surveyed cross section data should be provided at that location. It is very easy to use the automatic cross section interpolation to generate cross sections. But if these cross sections are not an adequate depiction of the actual geometry, you may be introducing

error into the calculation of the water surface profile. Whenever possible, use topographic maps to assist you in evaluating whether or not the interpolated cross sections are adequate. Also, once the cross sections are interpolated, they can be modified just like any other cross section.

If the geometry between two surveyed cross sections does not change linearly, then the interpolated cross sections will not adequately depict what is in the field. When this occurs, the modeler should either get additional surveyed cross sections, or adjust the interpolated sections to better depict the information from the topographic map.

River Ice

The current version of HEC-RAS allows the user to model ice-covered channels. This section of the users manual will describe how to enter the data describing the ice cover and the ice cover properties. If the ice cover geometry is known, that is, if the ice cover thickness and roughness are known throughout the reaches of interest, the user can supply these data and describe the ice cover directly. If the ice cover results from a wide-river type jam, HEC-RAS will estimate the jam thickness in reaches where the ice jam occurs. In this case, the user can supply the material properties of the jam or use the default values. To find out how to view specific results for a channel with an ice cover, see Chapter 9 of this User's manual.

Entering and Editing Ice Data

River ice data can be entered in two ways: by using the **Add Ice Cover** option under the **Options** Menu found at the top of the Cross Section Data Editor (Figure 6.2), or by using **Tables** Menu found at the top of the Geometric Data window (Figure 6.1). Both ways of entering data will be described below. It is important to remember that at least two cross sections are required to define the ice cover. A cross section should be placed at the upstream and downstream ends of each ice-covered reach.

Entering Ice Data at a Cross Section

To enter river ice data the user presses the **Cross Section** button on the Geometric Data window (Figure 6.1). Once the cross section button is pressed the Cross Section Data Editor will appear as shown in Figure 6.2. See the CROSS SECTION DATA section of the User's Manual, for information on selecting the appropriate river, reach, and cross section in the Cross Section Data Editor. Once a cross section with an ice cover has been selected, choose the "**Add ice cover...**" option under the **Options** Menu found at the top of the Cross Section Data Editor (Figure 6.2). This will open the Ice Cover Editor (Figure 6.52). All ice data for this cross section can be entered with this editor.

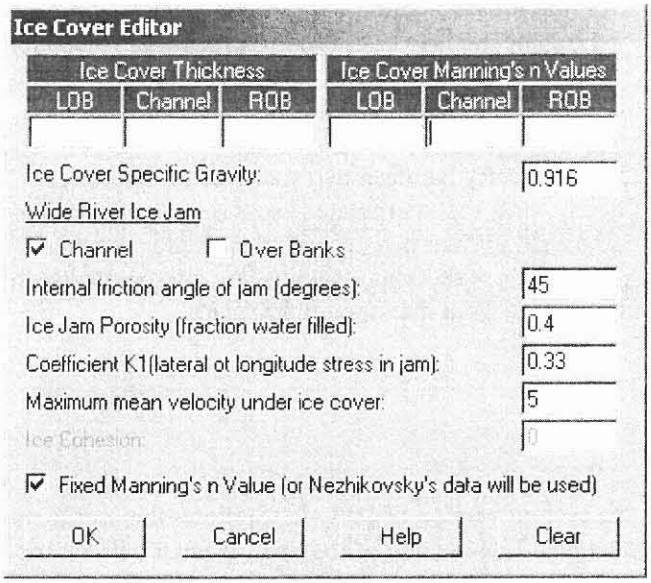


Figure 6.52 Ice Cover Editor

Ice Cover Thickness. The ice cover thickness in the left overbank (LOB), main channel (Channel), and right overbank (ROB), are entered here. If there is no ice in any of these areas, a thickness of zero should be entered.

Ice Cover Manning's n Values. The Manning's n value of the ice cover in the left overbank (LOB), main channel (Channel), and right overbank (ROB), are entered here. If any part of a cross section has a non-zero ice thickness, a Manning's n value must be supplied.

Ice Cover Specific Gravity. The default value is 0.916. The user can supply an alternative value here.

Wide River Ice Jam. The boxes under this option are checked if this section is to be treated as a wide river ice jam. In this case, HEC-RAS will estimate the jam thickness using the complete ice jam force balance as described in the Hydraulic Reference Manual. The user can confine the jam to the main channel or allow the jam to be in the channel and overbank areas by checking the proper boxes. If the ice cover is confined to the channel, the overbanks can have a known ice thickness (including an ice thickness of zero) assigned to them in the Ice Cover Thickness option. If the Wide River Ice Jam option is selected, an ice cover thickness must be supplied for the main channel using the **Ice Cover Thickness** Option or through the Ice Tables (see below). This ice cover thickness will be used as the initial estimate of the ice jam thickness and will also serve as the minimum thickness allowed for the ice jam at that section. If the jam is allowed in the overbank areas, the channel and overbanks hydraulic properties will be combined to calculate a single jam thickness for the channel and overbanks. **NOTE:** A wide river jam cannot be selected for an entire river channel. A cross section with fixed ice cover

geometry must be included at the upstream end and the downstream end of the wide river ice jam to serve as the boundary conditions for the jam. There is no limit to the number of separate wide river jams that can exist in a river network. However, every ice jam must have a cross section with fixed ice geometry at its upstream and downstream limit. Ice jams can extend through any number of junctions. However, the jam will only be extended between reaches that have identical reach names.

Internal Friction Angle of the Jam (degrees). This describes the "strength" of the ice jam as a granular material. The default value is 45_degrees.

Ice Jam Porosity (fraction water filled). This describes the fraction of the ice jam that is filled with liquid water. The default value is 0.4.

Coefficient K1 (longitudinal to lateral stress in jam). This describes the ratio of the lateral stress and the longitudinal stress in the jam. It is the efficiency of the jam in transferring longitudinal stress into lateral stress against the channel banks. The default value is 0.33

Maximum mean velocity under ice cover. This option limits the maximum mean velocity under a wide river ice jam. The default value is 5 fps. If the maximum mean velocity is greater than this, the ice cover will be thinned until the maximum velocity is attained, or the minimum ice thickness supplied by the user is reached. In any case, the jam thickness will not be allowed to be thinner than the user supplied thickness. This option prevents the jam from thickening to such an extent that the entire cross sectional area of the channel would become blocked.

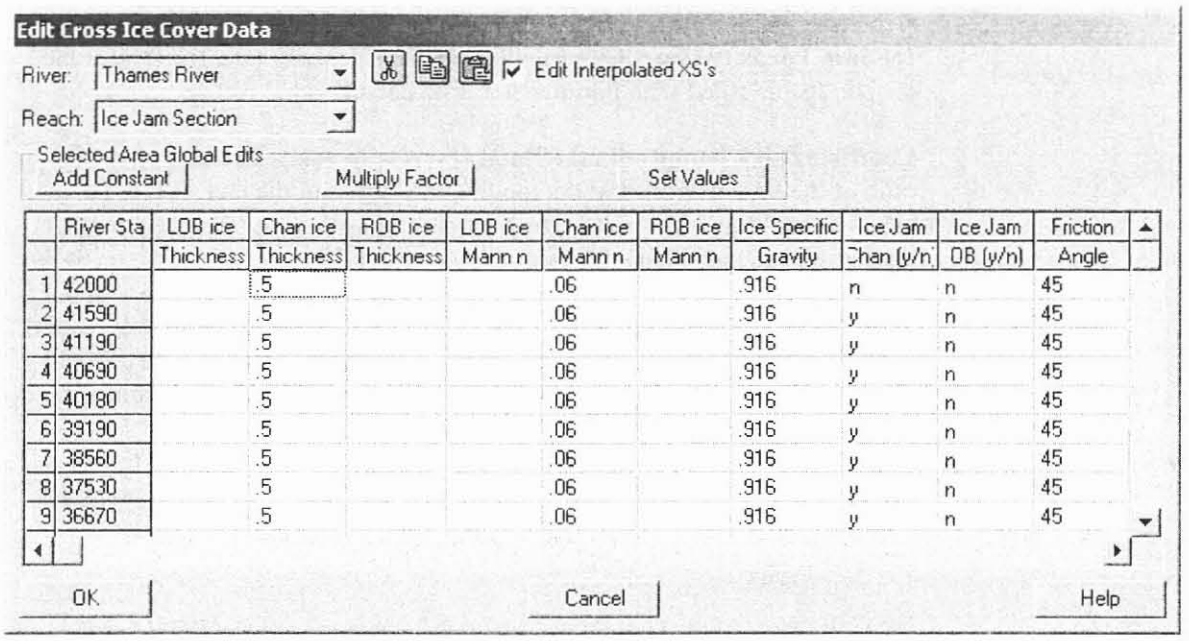
Ice Cohesion. At present, the ice jam cohesion is set to the default value of zero. This cannot be changed by the user. A value of zero is appropriate for breakup ice jams.

Fixed Manning's n Value (or Nezhikovsky's data will be used). The Manning's n value of the ice jam can be specified by the user or estimated using the empirical relationships developed from Nezhikovsky's data (1964). The empirical relationships estimate the Manning's n value on the basis of the jam thickness and the total water depth. The default is the user supplied Manning's n value.

Once all the ice data have been entered and edited, click the **OK** button. At the bottom of the Cross Section Data Editor, in the space entitled "List of special notes for cross section," the words "Ice cover" will now appear. The user can now click on the words "Ice cover" to return to the ice cover editor for that cross section.

Entering Ice Data Through a Table

Ice cover information can also be entered using the Tables Menu found at the top of the Geometric Data Window (Figure 6.1). To enter data the user selects the **Ice Cover** Option under the **Tables** Menu. All the information that can be entered under the Ice Cover Editor can also be entered using the Ice Cover table. It is often very convenient to enter and view data for more than one cross section at a time (Figure 6.53).



	River Sta	LOB ice Thickness	Chan ice Thickness	ROB ice Thickness	LOB ice Mann n	Chan ice Mann n	ROB ice Mann n	Ice Specific Gravity	Ice Jam Chan (y/n)	Ice Jam OB (y/n)	Friction Angle
1	42000		.5			.06		.916	n	n	45
2	41590		.5			.06		.916	y	n	45
3	41190		.5			.06		.916	y	n	45
4	40690		.5			.06		.916	y	n	45
5	40180		.5			.06		.916	y	n	45
6	39190		.5			.06		.916	y	n	45
7	38560		.5			.06		.916	y	n	45
8	37530		.5			.06		.916	y	n	45
9	36670		.5			.06		.916	y	n	45

Figure 6.53 Entering ice information using a Table

The user has the option of entering the ice thickness in the left overbank (LOB ice Thickness), the main channel (Chan ice Thickness), and the right overbank (ROB ice Thickness); the Manning's n value of the left overbank ice cover (LOB ice Mann n), the main channel ice cover (Chan ice Mann n), and the right overbank ice cover (ROB ice Mann n); and the specific gravity of the ice cover (Ice gravity). The user can also choose if the ice cover in the main channel is the result of a wide river ice jam (Ice Jam Chan. **Note: only y or n can be entered here**), and choose if the overbanks are also included in the wide river ice jam (Ice Jam OB. **Note: only y or n can be entered here**). The user can further select the internal friction angle of the ice jam (Friction Angle); the porosity of the ice jam (Porosity); the longitudinal to lateral stress ratio of the ice jam (Stress K1 ratio); the maximum allowable under ice flow velocity (Max Velocity); and if the Manning's n value of the ice jam is fixed, that is selected by the user, or if the Manning's n value will be determined by HEC-RAS (**Note: only y or n can be entered here**).

As in all instances where a Table is used to enter data, in each column the user has the option of entering one or more values, adding a constant to one

or more of the values, multiplying a group of values by a factor, or changing a group of values to a specific value. Additionally, cut, copy, and paste buttons are provided to pass data to and from the Windows Clipboard.

Entering Ice Data at Bridges

The influence of ice on the hydraulics of bridges is a relatively unstudied area. Little is known about the ways in which a wide river ice jam interacts with the various components of a bridge. The important components of a bridge that may interact with an ice jam include the piers, low chord, approaches, and deck. Previous investigations of ice jams in rivers with bridges have largely ignored their presence, arguing that observed ice jams did not contact the low steel significantly. Removing the bridge information for an ice jam study still remains an option. However to allow a user to efficiently use HEC-RAS with ice *and* with bridges, three separate options are provided. These options allow the user to selectively decide at each bridge whether or not the ice cover can interact with the structure. When modeling ice at bridges, users should carefully evaluate the results for consistency and accuracy.

Ice information at bridges is entered using the Bridge/Culvert editor found under the Geometry editor. Use the options menu in the Bridge/Culvert editor to select the ice option. This will open a window as shown in Figure 6.54.

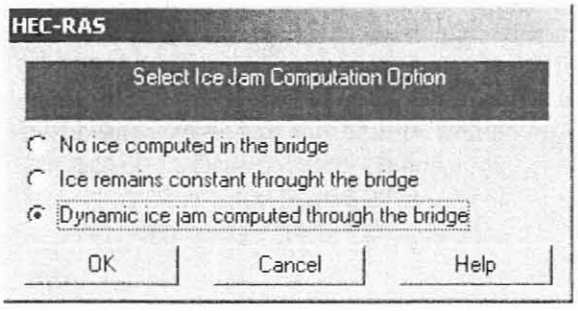


Figure 6.54 Entering ice information at bridges

No ice computed in the bridge. In this case no ice calculations will be preformed at the bridge itself and the ice thickness at the bridge will be assumed to be zero.

Ice remains constant through the bridge. In this case, the ice thickness at the cross section immediately upstream of the bridge will be used. If the ice thickness is calculated as a wide river jam, this thickness will be used.

Dynamic ice jam computed through the bridge. In this case, the wide river ice jam calculations will be preformed at the bridge cross section. The user must check for inconsistent results, especially if any part of the ice jam is above the low chord of the bridge.

Setting Tolerances for the Ice Jam Calculations

The user can override the default settings for the ice jam calculation tolerances which are used in the solution of the ice jam force balance equation. The tolerances are set as multiples of the *water surface calculation tolerance* used in the solution of the energy equation, described in the Simulation Options section of Chapter 7. The user can change the values of these tolerances by changing the *water surface calculation tolerance*. The tolerances are as follows:

Ice thickness calculation tolerance. This tolerance is compared with the difference between the computed and assumed ice thickness at a cross section. It is set to ten times the *water surface calculation tolerance.* Its default value is 0.1 ft.

Global ice thickness calculation tolerance. This tolerance is compared with the difference between the computed ice thickness at each cross section between successive solutions of the ice jam force balance equation and the energy equation. It is set to ten times the *water surface calculation tolerance.* Its default value is 0.1 ft.

Global water level calculation tolerance. This tolerance is compared with the difference between the computed water surface elevations at each cross section between successive solutions of the ice jam force balance equation and the energy equation. It is set to six times the *water surface calculation tolerance.* Its default value is 0.06 ft.

Maximum number of ice jam iterations. This variable defines the maximum number of times for successive solutions of the ice jam force balance equation and the energy equation. It is set to 2.5 times the *maximum number of iterations*. Its default value is 50.

Viewing and Editing Data Through Tables

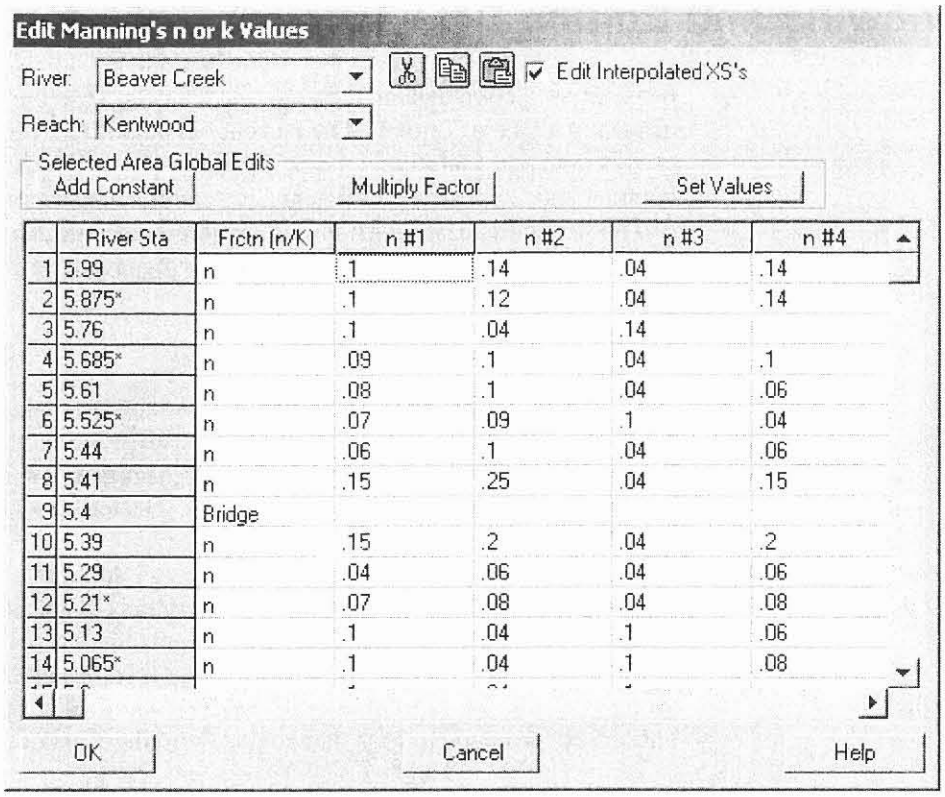
Once cross-section data are entered, the user can view and edit certain types of data in a tabular format. The current version of HEC-RAS allows the user to view and edit Manning's n or k values, cross-section reach lengths, contract and expansion coefficients, ice cover, cross-section river stationing, and node names. These options are available from the **Tables** menu option on the **Geometric Data** editor. The following is a description of each option.

Manning's n or k values

It is often desirable to view and edit the Manning's n values or roughness heights (k values) for several cross sections all at the same time. From the **Geometric Data** editor, the user can select **Manning's n or k values** from the **Tables** menu item. Once this option is selected, a window will appear as shown in Figure 6.55.

As shown in Figure 6.55, the user has the options of selecting either n or k values to be used as the roughness coefficient, add a constant to one or more of the n or k values, multiply a group of n or k values by a factor, or change a group of n or k values to a specific value. Additionally, cut, copy, and paste buttons are provided to pass data to and from the Windows Clipboard.

To add a constant to a group of n or k values, the user must first highlight the values that they would like to change. Highlighting is accomplished by placing the mouse in the upper left cell of the desired cells to highlight, then press the left mouse button and drag the cursor to the lower left corner of the desired cells to highlight. When the left mouse button is released, the cells that are selected will be highlighted (except the first cell). Once the user has highlighted the desired cells to be modified, press the **Add Constant** button. This will bring up a pop up window, which will allow the user to enter a constant value that will be added to all cells that are highlighted.



	River Sta	Frctn (n/K)	n #1	n #2	n #3	n #4
1	5.99	n	.1	.14	.04	.14
2	5.875*	n	.1	.12	.04	.14
3	5.76	n	.1	.04	.14	
4	5.685*	n	.09	.1	.04	.1
5	5.61	n	.08	.1	.04	.06
6	5.525*	n	.07	.09	.1	.04
7	5.44	n	.06	.1	.04	.06
8	5.41	n	.15	.25	.04	.15
9	5.4	Bridge				
10	5.39	n	.15	.2	.04	.2
11	5.29	n	.04	.06	.04	.06
12	5.21*	n	.07	.08	.04	.08
13	5.13	n	.1	.04	.1	.06
14	5.065*	n	.1	.04	.1	.08

Figure 6.55 Manning's n Data View and Editing Table

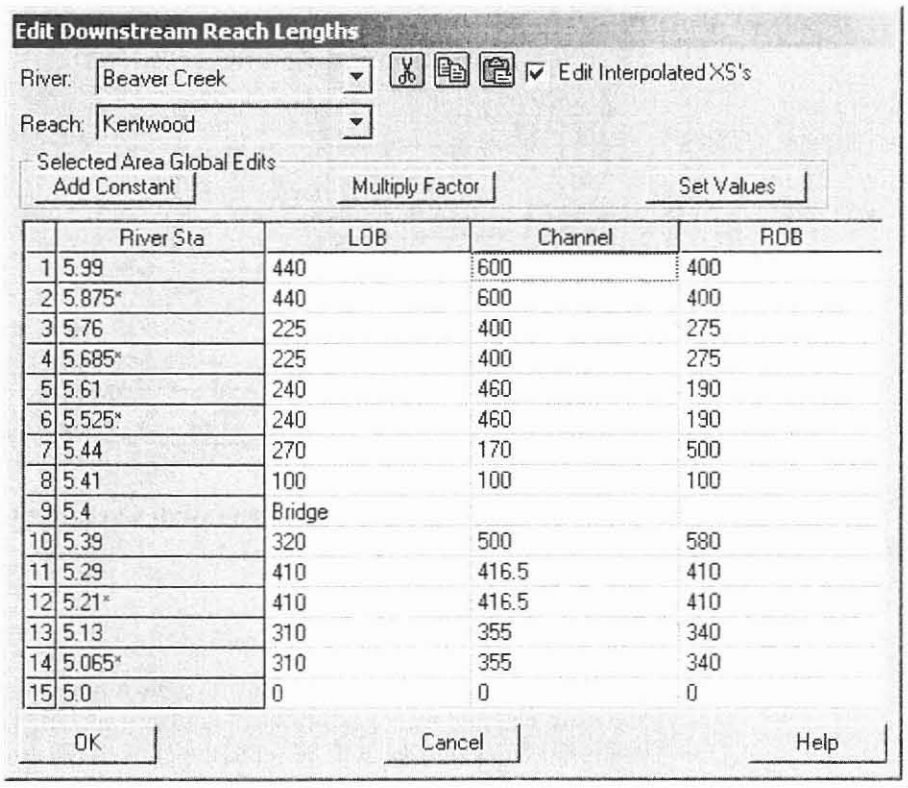
To multiply a group of n or k values by a factor, the user first highlights the desired cells. Once the cells are highlighted, pressing the **Multiply by a Factor** button will bring up a pop up window. This window allows the user to enter a value that will be multiplied by each of the highlighted cells.

To set a group of n or k values to the same number, the user must first highlight the values that they would like to change. Once the cells are highlighted, pressing the **Set Values** button will bring up a pop up window. This window will allow the user to enter a specific n or k value, which will replace all of the highlighted values.

The user can also go directly into the table and change any individual values.

Reach Lengths

The user has the ability to view and edit cross section reach lengths in a tabular format. This is accomplished by selecting **Reach Lengths** from the **Tables** menu of the Geometric Data editor. Once this option is selected, a window will appear as shown in Figure 6.56. The user has the same editing features as described previously for the n values table. See the discussion under Manning's n or k values, in the previous section, for details on how to edit the data.

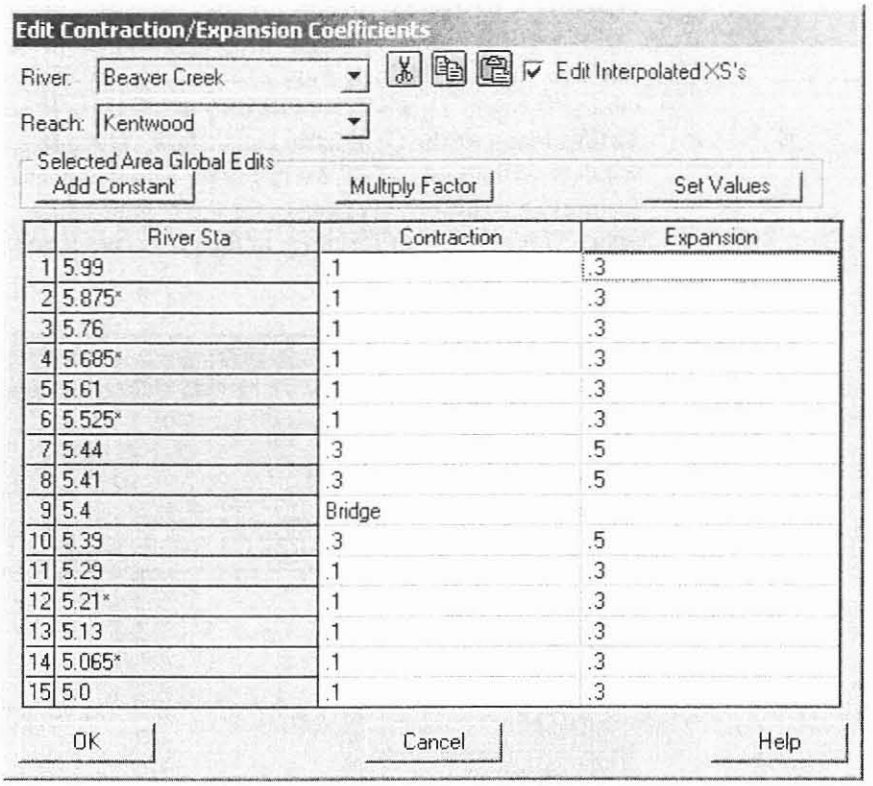


	River Sta	LOB	Channel	ROB
1	5.99	440	600	400
2	5.875*	440	600	400
3	5.76	225	400	275
4	5.685*	225	400	275
5	5.61	240	460	190
6	5.525*	240	460	190
7	5.44	270	170	500
8	5.41	100	100	100
9	5.4	Bridge		
10	5.39	320	500	580
11	5.29	410	416.5	410
12	5.21*	410	416.5	410
13	5.13	310	355	340
14	5.065*	310	355	340
15	5.0	0	0	0

Figure 6.56 Reach Lengths View and Editing Table

Contraction and Expansion Coefficients

The user has the ability to view and edit contraction and expansion coefficients in a tabular format. This is accomplished by selecting **Coefficients** from the **Tables** menu of the Geometric Data editor. Once this option is selected, a window will appear as shown in Figure 6.57. The user has the same editing features as described previously for the n values table. See the discussion under Manning's n values, in the previous section, for details on how to edit the data.



	River Sta	Contraction	Expansion
1	5.99	.1	.3
2	5.875*	.1	.3
3	5.76	.1	.3
4	5.685*	.1	.3
5	5.61	.1	.3
6	5.525*	.1	.3
7	5.44	.3	.5
8	5.41	.3	.5
9	5.4	Bridge	
10	5.39	.3	.5
11	5.29	.1	.3
12	5.21*	.1	.3
13	5.13	.1	.3
14	5.065*	.1	.3
15	5.0	.1	.3

Figure 6.57 Contraction and Expansion Coefficients Table

Bank Stations

This table allows the user to set or move the main channel bank stations. If the main channel bank stations have not been set, and the user brings up this table, the bank stations will be set to the ends of the cross section. If the bank stations are already set the user can adjust the bank stations by adding a constant, multiplying them by a factor, or setting them to a specific value. If the new bank stations do not exist in the cross section, the program will automatically interpolate them.

Levees

This table allows the user to easily enter and edit levee stations and elevations. The editor allows the user to quickly move levee stations and/or elevations. This can be very useful when trying to decide on a location for levee setbacks and their corresponding elevations.

Ice Cover

This option allows the user to enter ice cover data in a tabular form. A detailed discussion of ice cover information was presented earlier in this chapter.

River Stationing

This option allows the user to view and edit the cross section river stationing in a tabular form. This is accomplished by selecting **River Stations** from the **Tables** menu of the Geometric Data editor. Once this option is selected, a window will appear as shown in Figure 6.58. This table allows the user to change the river stationing of individual cross sections, add a constant value to the river stationing of selected cross sections (those cross sections highlighted by the user), multiply the selected cross sections river stationing by a factor, or to renumber the cross section river stationing based on the main channel reach lengths

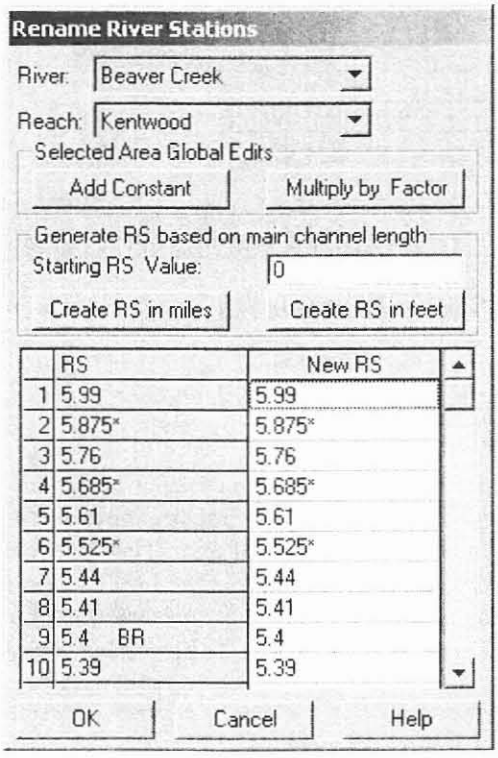


Figure 6.58 Cross Section River Stationing View and Editing Table

Node Names

This option allows the user to add an additional name to a node (a node is a cross section, bridge, culvert, inline structure, etc…). The name can be up to 16 characters long. The user can request that the name be displayed on a profile plot or on a cross-section plot. To use this feature, select **Node Names** from the **Tables** menu. When this option is selected a window will appear as shown in Figure 6-59. Enter any text name that you want at a desired location within the model.

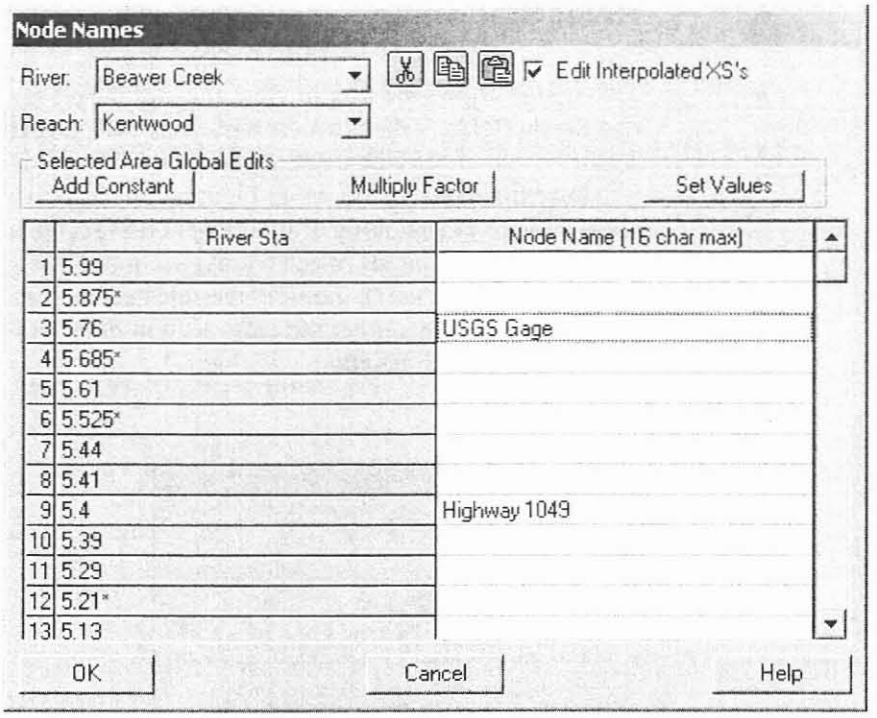


Figure 6.59 Node Name Table Editor

Node Descriptions

This table allows the user to enter a description for any node (cross section, bridge, culvert, inline structure, lateral structure, and pump stations). The description can be up to six rows of text. The table allows the user to display any number of the rows at one time. The user can request that the description be displayed on a profile plot or on a cross-section plot. To use this feature, select **Node Descriptions** from the **Tables** menu. Enter any text description that you want at a desired location within the model.

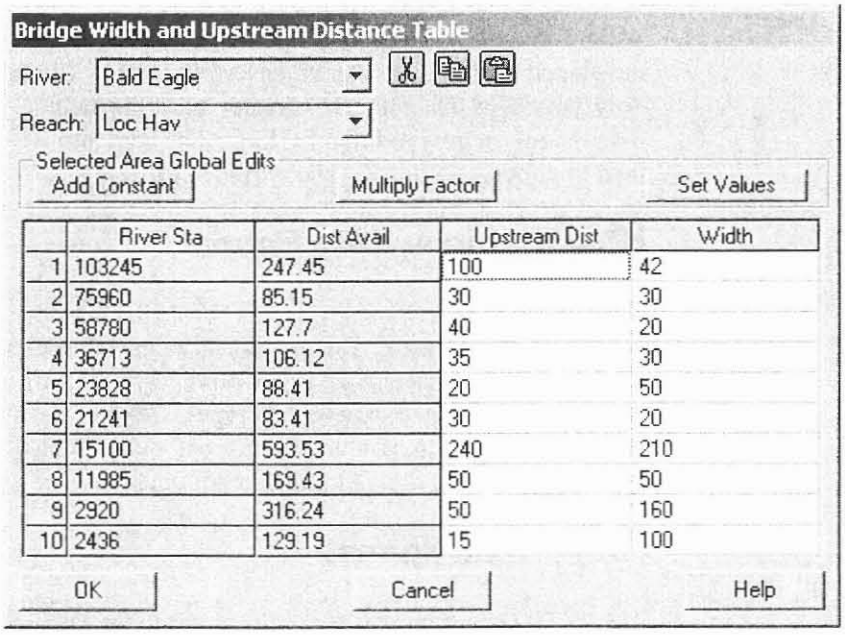
Picture File Associations

This table allows the user to see and edit the directories that pictures are attached to for the project. For example, if all of the pictures for a project were in a directory separate from the project, and these pictures were then moved to another directory, this table would allow you to easily change the associated directory for the pictures.

Bridge Width Table

This option allows the user to view and/or modify bridge width and distance data. Previous versions of HEC-RAS (versions 2.21 and earlier) allowed the user to enter a zero length between the cross sections inside of a bridge and the cross sections just outside of the bridge. This creates and unrealistic water surface profile in the vicinity of the bridge. Current versions require

the user to maintain some distance between the outside cross sections and the bridge structure. This table was added to make the process of modifying old data sets less painful. When this option is selected, a window will appear as shown in Figure 6-60. As shown in Figure 6-60, the user is given the length between the cross sections that bound the bridge, the distance between the upstream cross section and the bridge, the bridge width, and the distance between the downstream cross section and the bridge. The user must ensure that the upstream and downstream distances are greater than zero. This will require entering an upstream distance, and then changing the bridge width to allow for a positive downstream distance.



	River Sta	Dist Avail	Upstream Dist	Width
1	103245	247.45	100	42
2	75960	85.15	30	30
3	58780	127.7	40	20
4	36713	106.12	35	30
5	23828	88.41	20	50
6	21241	83.41	30	20
7	15100	593.53	240	210
8	11985	169.43	50	50
9	2920	316.24	50	160
10	2436	129.19	15	100

Figure 6.60 Bridge Width and Distance Table

Weir and Gate Coefficient Table

This table allows the user to see and edit all of the weir and gate coefficients for all of the inline and lateral structures within the model.

Importing Geometric Data

HEC-RAS has the ability to import geometric data in several different formats. These formats include: a GIS format (developed at HEC); the USACE Standard Surveyor format; HEC-2 data format; HEC-RAS data format; UNET geometric data format; and the MIKE11 cross section data format. Data can be imported into an existing HEC-RAS geometry file or for a completely new geometry file. Multiple data files can be imported into the same geometric data file on a reach-by-reach basis.

To import data into a HEC-RAS geometric data file, the user selects the **Import Geometric Data** option from the **File** menu of the Geometric Data window. Once this option is selected, the user then selects one of the three available formats from the list. After a format is selected, the user will be asked if they want to add the data to the current geometry file, or if they want to clear the current geometry file before importing the data. Once this choice is made, the user will be prompted to enter the name of the file containing the data. The following is a discussion of each of the three file formats.

GIS Format

A file format for interfacing HEC-RAS with GIS/CADD systems has been developed at HEC. A detailed description of the file format is contained in Appendix B of this manual. Chapter 14 of this manual provides detailed discussions on how to import GIS/CADD data into HEC-RAS, as well as how to export computed water surface profiles back to GIS/CADD systems.

USACE Survey Data Format

The U.S. Army Corps of Engineers (USACE) has developed a standard file format for survey data. This format is documented in Chapter 6 of Engineering Manual (EM) 1110-1-1005. The USACE survey format encompasses a wide range of data types. The current version of HEC-RAS has the capability to read this file format, but only cross section data are extracted from the file. At this time all other data are ignored.

HEC-2 Data Format

The HEC-2 program was the predecessor to the HEC-RAS software package. The HEC-2 program was used for many years to compute steady flow water surface profiles. Consequently, thousands of data sets exist in the HEC-2 data format. HEC-RAS has two ways of importing HEC-2 data. The first way is accomplished through the use of the **Import HEC-2 Data** option from the **File** menu on the main HEC-RAS window. When this method is used, it is assumed that the user has started a new project; and therefore all of the HEC-2 data is imported (geometric data, flow data, and plan information). A second way of importing HEC-2 data is provided from the geometric data editor. This way of importing HEC-2 data allows the user to bring the data into existing HEC-RAS geometric data files. This method also allows the user to import multiple HEC-2 data files into the same HEC-RAS geometric data file. However, when importing HEC-2 data from the geometric data window, only the geometric data contained in the HEC-2 files will be imported. All of the other data (flow data and plan information) will be ignored.

HEC-RAS Data Format

This option allows the user to combine several HEC-RAS geometry files into a single geometry file. For example, if several pieces of a river system were developed as separate HEC-RAS models, this option could be used to put

them together into one model.

UNET Geometric Data Format

This option allows the user to import a UNET geometric data file (CSECT geometry file). UNET is an unsteady flow program developed by Dr. Robert Barkau. The Corps, as well as many other agencies, has used this software for many years. UNET models are often very complex, consisting of many river reaches that can be connected in numerous ways. The HEC-RAS UNET importer does not have enough information to draw the schematic in the proper manner. The river reaches and storage areas will be connected correctly, but the user will need to edit the schematic to make it look like the actual river system.

MIKE11 Cross-Section Data

This option allows the user to import cross section data from the MIKE11 program. MIKE11 is a one-dimensional river hydraulics model developed by the Danish Hydraulic Institute. Users must first export the MIKE11 data to a raw text file. This is an available option from MIKE11. Once the data is in the text file format, it can be imported into HEC-RAS.

Geometric Data Tools

Several tools are available from the Geometric Data editor to assist you in the development and editing of data. These tolls consist of: cross section interpolation; channel modification; graphical cross section editor; reverse stationing data; set ineffective flow areas to permanent mode; cross section points filter; fixed sediment elevation; pilot channels; and GIS cut line check. The cross section interpolation tool has been described previously in this chapter. Channel modification is described separately in Chapter 13 of this manual. The following is a short description of each of the tools.

Graphical Cross Section Editor

A graphical cross section editor is available from the **Tools** menu of the Geometric Data Editor window. When this option is selected, a window will appear as shown in Figure 6.61.

The user has the option to move objects (objects are ground points, main channel bank stations, ineffective flow areas, levees, and blocked obstructions), delete objects, or add new objects. To move an object, the user first selects **Move Objects** from the **Options** menu. Then move the mouse pointer over the object that you want to move, press down the left mouse button, and then move the object. When you are finished moving the object, simply release the left mouse button and the object will be moved. To delete

CHAPTER 7

Performing a Steady Flow Analysis

This chapter discusses how to calculate steady flow water surface profiles. The chapter is divided into two parts. The first part discusses how to enter steady flow data and boundary conditions. The second part discusses how to develop a plan and perform the calculations.

Contents

- Entering and Editing Steady Flow Data
- Performing Steady Flow Calculations

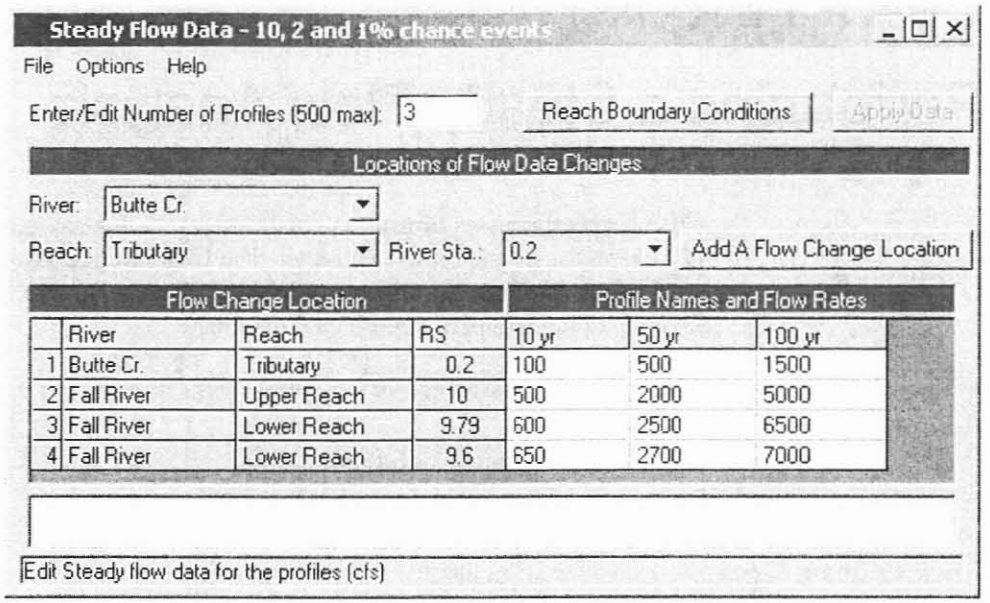
Entering and Editing Steady Flow Data

Once all of the geometric data are entered, the modeler can then enter any steady flow data that are required. To bring up the steady flow data editor, select **Steady Flow Data** from the **Edit** menu on the HEC-RAS main window. The steady flow data editor should appear as shown in Figure 7.1

Steady Flow Data

The user is required to enter the following information: the number of profiles to be calculated; the peak flow data (at least one flow for every river reach and every profile); and any required boundary conditions. The user should enter the number of profiles first. The next step is to enter the flow data. Flow data are entered directly into the table. Use the mouse pointer to select the box in which to enter the flow then type in the desired value.

Flow 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 reach. The flow data can be changed at any cross section within a reach. To add a flow change location to the table, first select the reach in which you would like to change the flow (from the river and reach boxes above the table). Next, select the River Station location for which you want to enter a flow change. Then press the **Add Flow Change Location** button. The new flow change location will appear in the table.



	River	Reach	RS	10 yr	50 yr	100 yr
1	Butte Cr.	Tributary	0.2	100	500	1500
2	Fall River	Upper Reach	10	500	2000	5000
3	Fall River	Lower Reach	9.79	600	2500	6500
4	Fall River	Lower Reach	9.6	650	2700	7000

Figure 7.1 Steady Flow Data Editor

Each profile is automatically assigned a title based on the profile number, such as profile #1 is assigned a title of "Prof #1," profile #2 is assigned a title of "Prof #2," etc. The user can rename the title for each profile by simply going into the options menu and selecting **Edit Profile Names**. Once this option is selected, a dialog will appear allowing you to rename each of the profile titles.

Boundary Conditions

After all of the flow data have been entered into the table, the next step is to enter any boundary conditions that may be required. To enter boundary conditions data, press the **Boundary Conditions** button at the top right of the steady flow data editor. The boundary conditions editor should appear as shown in Figure 7.2.

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.

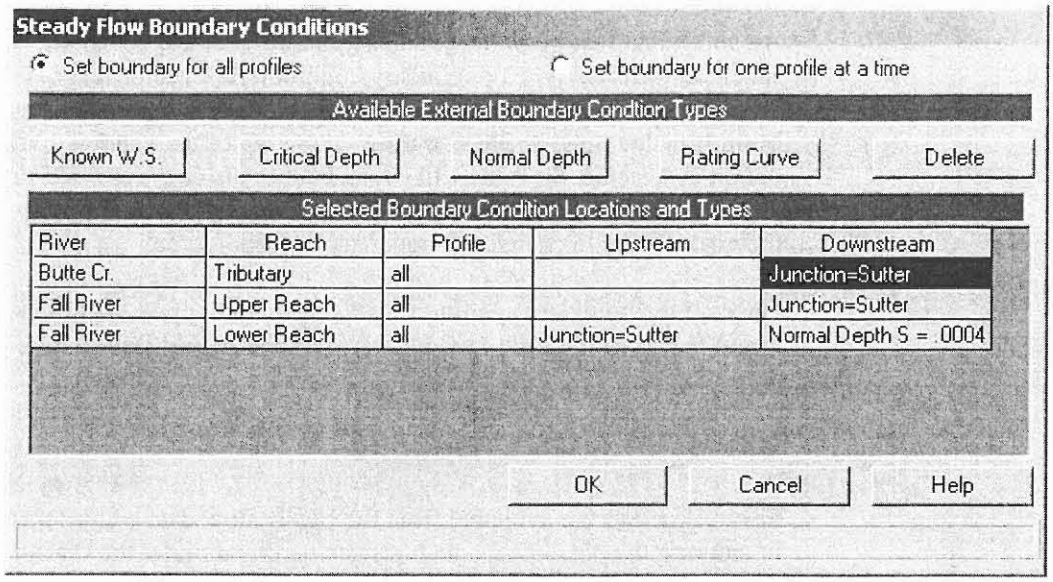


Figure 7.2 Steady Flow Boundary Conditions Editor

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 only required to enter the necessary external boundary conditions.

To enter a boundary condition, first use the mouse pointer to select the cell location in which you would like to enter a boundary condition. Then select the type of boundary condition from the four available types listed above the table. The four types of boundary conditions consist of:

Known Water Surface Elevations - For this boundary condition the user must enter a known water surface for each of the profiles to be computed.

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.

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. If the energy slope is unknown, the user could approximate it by entering either the slope of the water surface of the slope of the channel bottom.

Rating Curve - When this type of boundary condition is selected, a pop up window appears allowing the user to enter an elevation versus flow rating curve. For each profile, the elevation is interpolated from the rating curve given the flow.

An additional feature of the boundary condition editor is that the user can specify a different type of boundary condition for each profile at the same location. This is accomplished by first selecting the option that says **"Set boundary for one profile at a time"** at the top of the window. When this option is selected, the table will expand out to provide a row for each profile, at every location. The user can then select the location and profile for which they would like to change the boundary condition type.

Once all the boundary conditions data are entered, press the **OK** button to return to the steady flow data editor. Press the **Apply Data** button to have the data accepted.

Steady Flow Data Options

Several options are available from the steady flow data editor to assist users in entering the data. These features can be found under the **Options** menu at the top of the window. The following options are available:

Undo Editing. This option allows the user to retrieve the data back to the form that it was in the last time the Apply Data button was pressed. Each time the Apply Data button is pressed, the Undo Editing feature is reset to the current information.

Delete Row From Table. This option allows the user to delete a row from the flow data table. To use this option, first select the row to be deleted with the mouse pointer. Then select **Delete Row From Table** from the options menu. The row will be deleted and all rows below it will move up one.

Delete All Rows From Table. This option allows the user to delete all of the rows from the table. To use this option, select **Delete All Rows From Table** from the **Options** menu. When this option is selected a window will appear with a question to make sure that deleting all of the row is what you really want to do.

Delete Column (Profile) From Table. This option allows the user to delete a specific column (profile) of data from the table. To use this option, first select the column that you want to delete by placing the mouse over any cell of that column and clicking the left mouse button. Then select **Delete Column (Profile) From Table** from the **Options** menus. The desired column will then be deleted.

Ratio Selected Flows. This option allows the user to multiply selected values in the table by a factor. Using the mouse pointer, hold down the left mouse button and highlight the cells that you would like to change by a factor. Next, select **Ratio Selected Flows** from the options menu. A pop up window will appear allowing you to enter a factor to multiply the flows by. Once you press the **OK** button, the highlighted cells will be updated with the new values.

Edit Profile Names. This option allows the user to change the profile names from the defaults of PF#1, PF#2, etc.

Set Changes in WS and EG. This option allows the user to 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. When this option is selected, a window will appear as shown in Figure 7.3. As shown, there are four options that the user can select from: **Additional EG**, **Change in EG**, **Known WS**, and **Change in WS**. The **Additional EG** option allows the user to add an 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. The **Change in EG** option allows the user to set a specific amount of energy loss between two cross sections. When this option is selected, the program does not perform an energy balance, it simply adds the specified energy loss to the energy of the downstream section and computes a corresponding water surface. The **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. The program will use the known water surface elevation and then move to the next section. The **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.

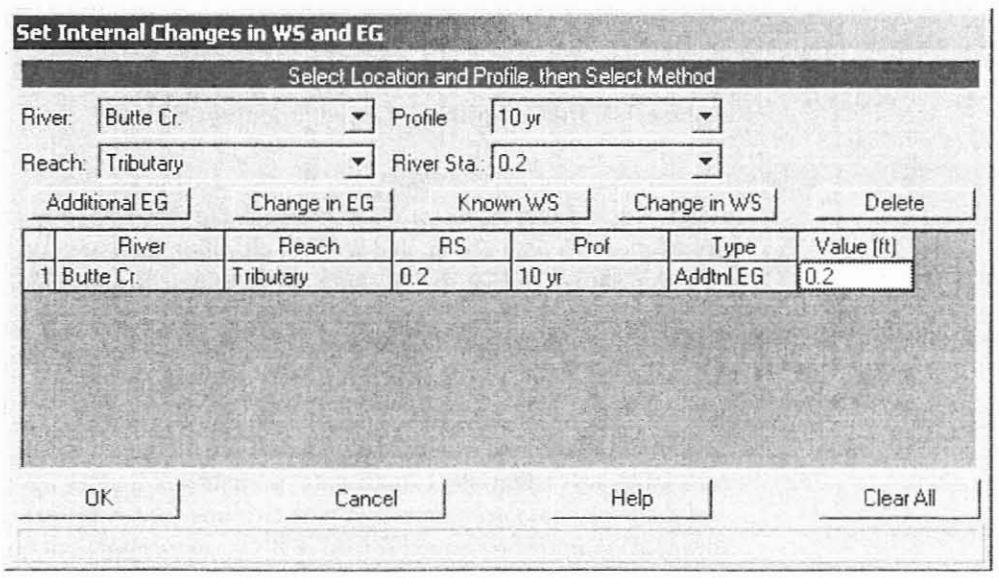


Figure 7.3 Setting Changes in Water Surface and Energy

As shown in Figure 7.3, to use the "Set Internal Changes in WS and EG" option, the user first selects the river, reach, river station, and profile that they would like to add an internal change too. Once the user has established a

location and profile, the next step is to select one of the four available options by pressing the appropriate button. Once one of the four buttons are pressed, a row will be added to the table at the bottom, and the user can then enter a number in the value column, which represents the magnitude of the internal change.

Observed WS. This option allows the user to enter observed water surfaces at any cross section for any of the computed profiles. The observed water surfaces can be displayed on the profile plots and in the summary output tables.

Gate Openings. This option allows the user to control gate openings for any inline or lateral gated spillways that have been added to the geometric data. When this option is selected, a window will appear as shown in Figure 7.4.

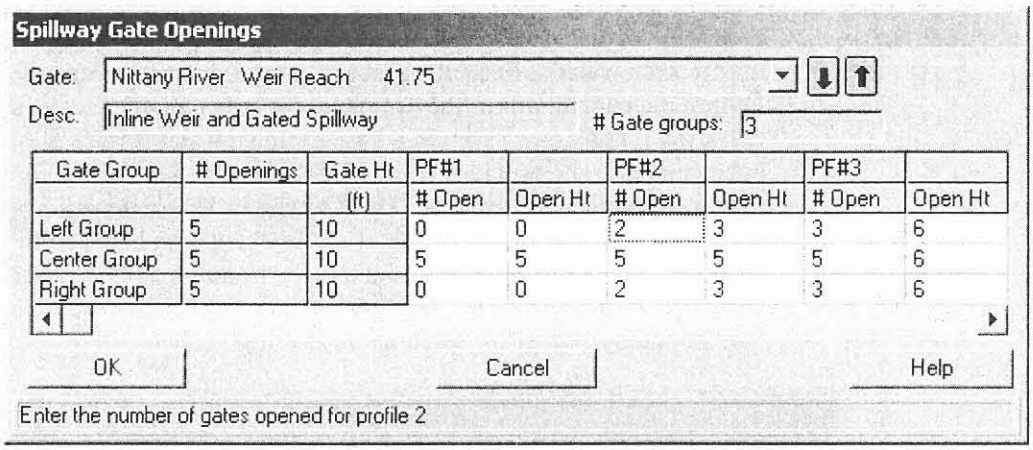


Figure 7.4 Inline Spillway Gated Openings Editor

As shown in Figure 7.4, for each profile the user can specify how many gates are opened per gate group, and at what elevation they are opened too. For the example shown in Figure 7.4, there are three gate groups labeled "Left Group," "Center Group," and "Right Group." Each gate group has five identical gate openings. All of the gate openings have a maximum opening height of ten feet. For profile number 1, only the middle gate group is opened, with all five gates opened to a height of five feet. For the second profile, all three gate groups are opened. The Left gate group has two gates opened to seven feet, the Center gate group has five gates opened to four feet, and the Right gate group has two gates opened for seven feet. This type of information must be entered for all of the profiles being computed.

Optimize Gate Openings. This option allows the user to have the program compute a gate setting at a structure in order to obtain a user specified water surface upstream of the structure. Given a user entered flow and upstream stage for each profile, the program will iterate with different gate settings until the desired upstream water surface is obtained. This option is very handy when modeling dams and reservoirs.

Initial Lateral Split Flow Values. This option allows the user to enter initial estimates of the flow that is leaving the main river through a lateral weir/spillway. Flow values can be entered for each profile. When a value is entered for this option, that amount of flow is subtracted from the main river before the first profile is computed. This option can be useful in reducing the required computation time, or allowing the program to reach a solution that may not otherwise been obtainable.

Storage Area Elevations. This option allows the user to enter water surface elevations for storage areas that have been entered into the geometric data. Storage areas are most often used in unsteady flow modeling, but they may also be part of a steady flow model. When using storage areas within a steady flow analysis, the user is requires to enter a water surface elevation for each profile.

Saving The Steady Flow Data

The last step in developing the steady flow data is to save the information to a file. To save the data, select the **Save Flow Data As** from the **File** menu on the steady flow data editor. A pop up window will appear prompting you to enter a title for the data.

Importing Data From The HEC Data Storage System (HEC-DSS)

HEC-DSS is a data base system that was specifically designed to store data for applications in water resources. The HEC-DSS system can store almost any type of data, but it is most efficient at storing large blocks of data (e.g., time-series data). These blocks of data are stored as records in HEC-DSS, and each record is given a unique name called a "pathname." A pathname can be up to 391 characters long and, by convention, is separated into six parts. The parts are referenced by the letters A, B, C, D, E, and F, and are delimited by a slash "/" as follows:

/A/B/C/D/E/F/

The pathname is used to describe the data in enough detail that various application programs can write to and read data from HEC-DSS by simply knowing the pathname. For more information about HEC-DSS, the user is referred to the "HEC-DSS, User's Guide and Utility Manuals" (HEC, 1995).

Many of the HEC application programs have the ability to read from and write to the HEC-DSS. This capability facilitates the use of observed data as well as passing information between software programs. The ability to read data from HEC-DSS has been added to HEC-RAS in order to extract flow and stage data for use in water surface profile calculations. It is a common practice to use a hydrologic model (i.e., HEC-HMS) to compute the runoff from a watershed and then use HEC-RAS to compute the resulting water surface profiles.

Reading data from HEC-DSS into HEC-RAS is a two-step process. First, the user must establish connections between HEC-RAS cross-section locations and pathnames contained in the HEC-DSS file. These connections are established by selecting the "**Set Locations for DSS Connections**" option from the **File** menu of the **Steady Flow Data** editor. When this option is selected, a window will appear as shown in Figure 7.5. The user selects cross-section locations for DSS connections by selecting a River, Reach, and River Station, then pressing the "Add selected location to table" button. When this button is pressed, a new row will be added to the table at the top of the window. The user should do this for all the locations where they want to establish connections to HEC-DSS data.

The next step is to open a particular HEC-DSS file. The user has the option of either typing the filename in directly, or using the open button, which is right next to the filename field. Once a DSS file is selected, a listing of the pathnames for all of the data contained in that file will appear in the table at the bottom of the window. The user can establish connections to more than one DSS file if desired.

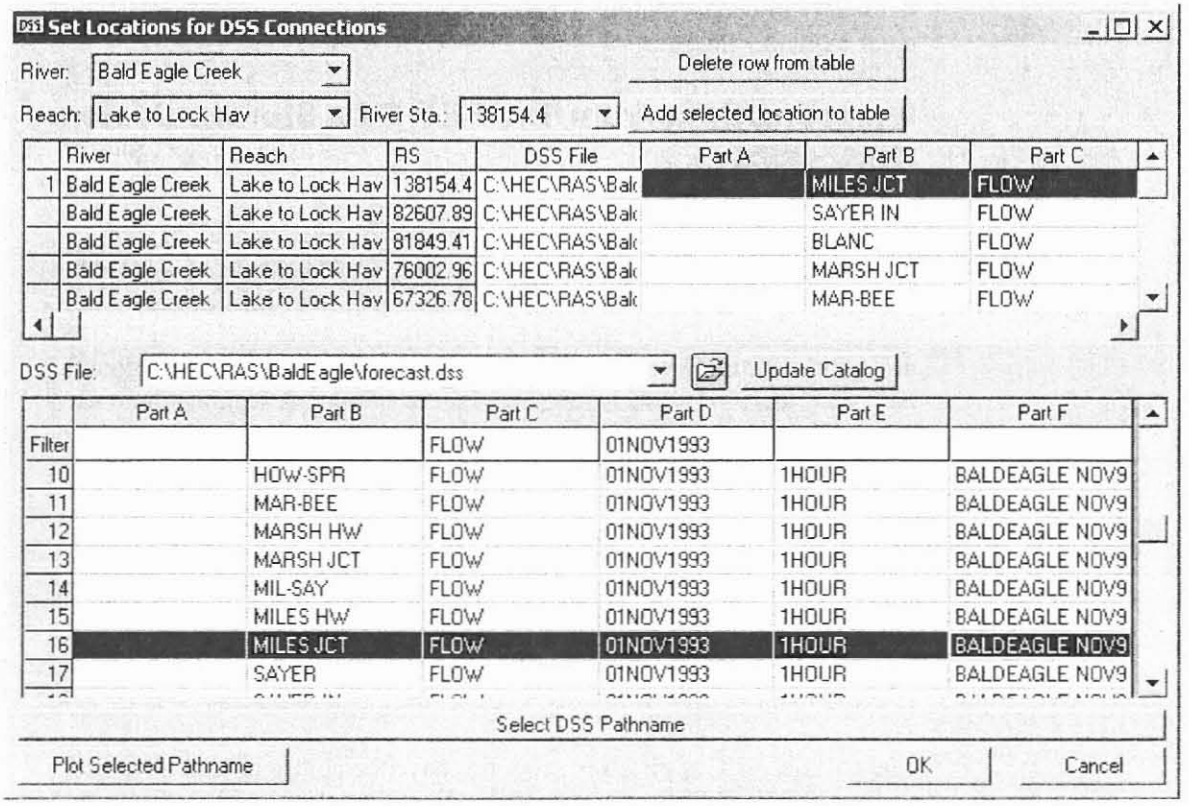


Figure 7.5 Editor for setting connections to HEC-DSS pathnames.

To establish the connection between an HEC-RAS cross section and a particular pathname in the DSS file, the user selects the row in the upper table that contains the river station that they want to connect data to. Next, they select the pathname that they want to connect to that river station from the

lower table. Finally, they press the button labeled "**Select DSS Pathname**," and the pathname is added to the table at the top of the window.

To make it easier to find the desired pathnames, a set of pathname part filters were added to the top row of the lower table. These filters contain a list of all the DSS pathname parts contained within the currently opened DSS file. If the user selects a particular item within the list of one of the pathname parts, then only the pathnames that contain that particular pathname part will be displayed. These filters can be used in combination to further reduce the list of pathnames displayed in the table. When a particular filter is left blank, that means that pathname part is not being filtered.

Another feature on the editor to assist in selecting the appropriate pathnames is the "Plot Selected Pathname" button. This button allows the user to get a plot or a table of the data contained within any record in the DSS file. The user simply selects a DSS pathname, and then presses the **Plot Selected Pathname** button, and a new window will appear with a graphic of the data contained within that record.

Once all of the pathname connections are set, the user presses the **OK** button to close the editor. The next step is to import the data. This is accomplished by selecting "**DSS Import**" from the **File** menu of the **Steady Flow Data** editor. When this option is selected, a window will appear as shown in Figure 7.6

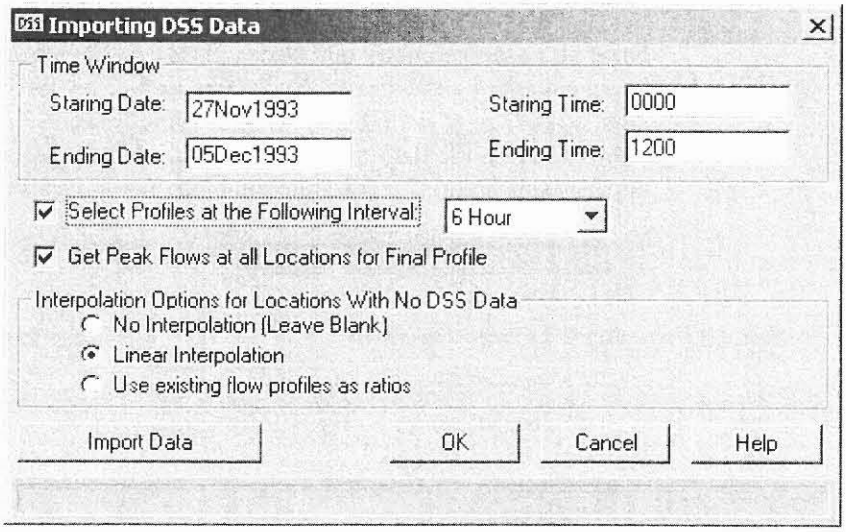


Figure 7.6 DSS Data Import Window

First the user sets a time window, which consists of a starting date and time and an ending date and time. When data are extracted from DSS, the program will only look at the data that is contained within the user specified time window.

Below the time window there are two options for selecting flow data to be extracted from the DSS file. The first option allows the user to pick off flow data at a specified time interval, starting with the beginning of the time window for the first profile. The second and subsequent profiles would be based on adding the user specified time interval to the start time of the time window. Flow data is extracted from the hydrographs at each of the locations being read from DSS. The second option listed on the window allows the user to get an overall peak flow for a profile computation. When this option is selected, the peak flow will be extracted from each hydrograph, within the time window specified. These peak flows will be made into the final profile in the flow data editor.

The bottom portion of the window contains options for interpolating flow data at locations that do not have hydrographs in the DSS file. After the flow data are read in, it will be necessary to interpolate flow data at all of the locations listed in the flow data editor that do not have values in the DSS file. Three options are available: no interpolation, linear interpolation, or using the flow data from an existing profile to calculated ratios for interpolating between points that have data. Once all the options are set, the user presses the "Import Data" button, to have the data imported and fill out the flow data editor.

Performing Steady Flow Calculations

Once all of the geometry and steady flow data have been entered, the user can begin calculating the steady flow water surface profiles. To perform the simulations, go to the HEC-RAS main window and select **Steady Flow Analysis** from the **Run** menu. The Steady Flow Analysis window will appear as in Figure 7.7 (except yours may not have a Plan title and short ID).

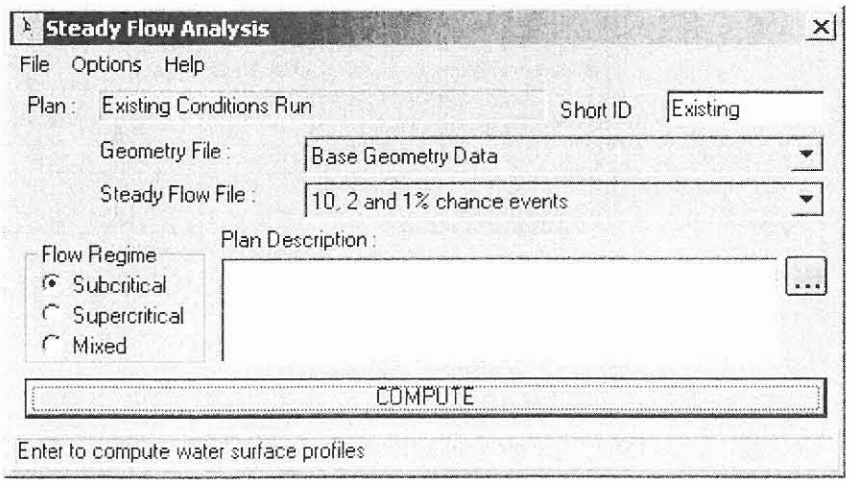


Figure 7.7 Steady Flow Analysis window

Defining A Plan

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

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

To establish a Plan, select **New Plan** from the **File** menu on the steady flow analysis window. When **New Plan** is selected, a window will appear prompting you to enter a title for the plan. After you enter the title, press the **OK** button to close the window and accept the title. The user will also be prompted to enter a short identifier for the plan. The short identifier is limited to 12 characters. It is very important to enter a short identifier that is descriptive of the plan. When viewing multiple plan output from the graphics and tables, the Short ID will be used to identify each plan.

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

Saving The Plan Information

To save Plan information to the hard disk, select **Save Plan** from the **File** menu of the simulation window. Whenever options are changed or modified on the Steady Flow Analysis window, the user should Save the Plan.

Simulation Options

The following is a list of the available simulation options under the **Options** menu of the Steady Flow Analysis window:

Encroachments. This option allows the user to perform a floodway encroachment analysis. For a detailed description of how to use the floodway encroachment capabilities of HEC-RAS, see Chapter 9 of the User's Manual (this manual). For a description of how the encroachment calculations are performed for the various encroachment methods, see Chapter 9 of the Hydraulic Reference Manual.

Flow Distribution Locations. This option allows the user to specify locations in which they would like the program to calculate flow distribution output. The flow distribution option allows the user to subdivide the left overbank, main channel, and right overbank, for the purpose of computing additional hydraulic information.

As shown in Figure 7.8, the user can specify to compute flow distribution information for all the cross sections (this is done by using the Global option) or at specific 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. The user can define up to 45 total slices. Each flow element (left overbank, main channel, and right overbank) must have at least one slice. The flow distribution output will be calculated for all profiles in the plan during the computations.

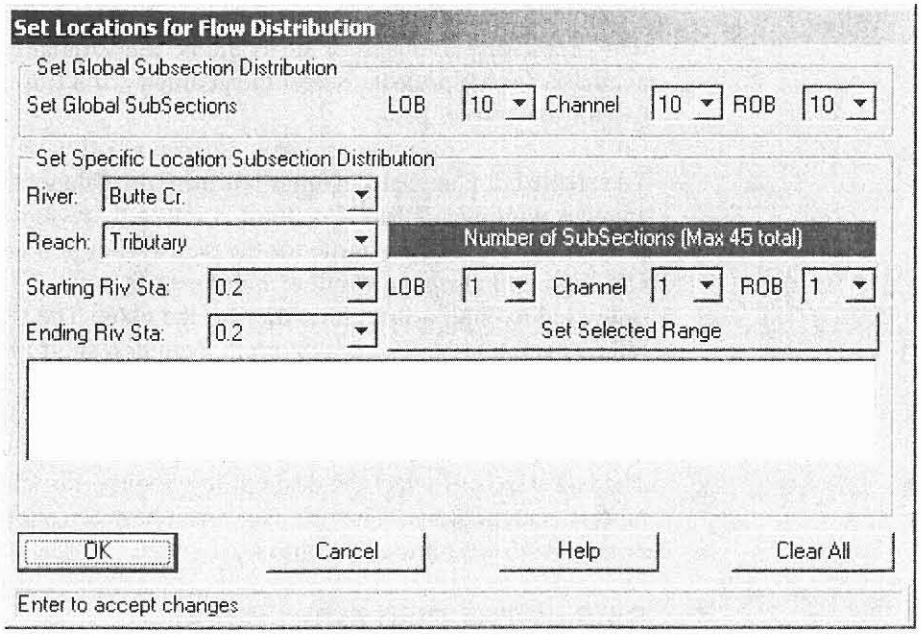


Figure 7.8 Window for Specifying the Locations of Flow Distribution

To set the flow distribution option for all the cross sections, simply select the number of slices for the left overbank, main channel, and right overbank from the **Set Global Subsections** portion of the window. To set flow distribution output at specific locations, use the **Set Specific Location Subsection Distribution** option.

During the normal profile computations, at each cross section where flow distribution is requested, the program will calculate the flow, area, wetted perimeter, percentage of conveyance, and average velocity for each of the user defined slices. For details on how the flow distribution output is actually calculated, see Chapter 4 of the HEC-RAS Hydraulic Reference Manual. For information on viewing the flow distribution output, see Chapter 9 of the User's Manual (this manual).

Conveyance Calculations. This option allows the user to tell the program how to calculate conveyance in the overbanks. Two options are available. The first option, **At breaks in n values only**, instructs the program to sum wetted perimeter and area between breaks in n values, and then to calculate conveyance at these locations. If n varies in the overbank the conveyance values are then summed to get the total overbank conveyance. The second option, **Between every coordinate point (HEC-2 style)**, calculates wetted perimeter, area, and conveyance between every coordinate point in the overbanks. The conveyance values are then summed to get the total left overbank and right overbank conveyance. These two methods can provide different answers for conveyance, and therefore different computed water surfaces. The **At breaks in n values only** method is the default.

Friction Slope Methods. This option allows the user to select one of four available friction slope equations, or to allow the program to select the method based on the flow regime and profile type. The four equations are:

- Average Conveyance (Default)

- Average Friction Slope

- Geometric Mean Friction Slope

- Harmonic Mean Friction Slope

Set Calculation Tolerances. This option allows the user to override the default settings for the calculation tolerances. These tolerances are used in the solution of the energy equation. **Warning !!!** - Increasing the default calculation tolerances could result in computational errors in the water surface profile. The tolerances are as follows:

Water surface calculation tolerance: This tolerance is used to compare against the difference between the computed and assumed water surface elevations. When the difference is less than the tolerance, the program assumes that it has a valid numerical solution.

Critical depth calculation tolerance: This tolerance is used during the critical depth solution algorithm.

Maximum number of iterations: This variable defines the maximum number of iterations that the program will make when attempting to balance a water surface.

Maximum difference tolerance: This tolerance is used during the balance of the energy equation. As the program attempts to balance the energy equation, the solution with the minimum error (assumed minus computed water surface) is saved. If the program goes to the maximum number of iterations without meeting the specified calculation tolerance, the minimum error solution is checked against the maximum difference tolerance. If the solution at minimum error is less than this value, then the program uses the minimum

error solution as the answer, issues a warning statement, and then proceeds with the calculations. If the solution at minimum error is greater than the maximum difference tolerance, then the program issues a warning and defaults the solution to critical depth. The computations then proceed from there.

Flow Tolerance Factor: This factor is only used in the bridge and culvert routines. The factor is used when the program is attempting to balance between weir flow and flow through the structure. The factor is multiplied by the total flow. The resultant is then used as a flow tolerance for the balance of weir flow and flow through the structure.

Maximum Iteration in Split Flow: This variable defines the maximum number of iterations that the program will use during the split flow optimization calculations.

Flow Tolerance Factor in Weir Split Flow: This tolerance is used when running a split flow optimization with a lateral weir/gated spillway. The split flow optimization continues to run until the guess of the lateral flow and the computed value are within a percentage of the total flow. The default value for this is 2 percent (.02).

Maximum Difference in Junction Split Flow: This tolerance is used during a split flow optimization at a stream junction. The program continues to attempt to balance flow splitting from one reach into two until the energy gradelines of the receiving streams are within the specified tolerance.

Each of these variables has an allowable range and a default value. The user is not allowed to enter a value outside of the allowable range.

Critical Depth Output Option. This option allows the user to instruct the computational program to calculate critical depth at all locations.

Critical Depth Computation Method. This option allows the user to select between two methods for calculating critical depth. The default method is the **Parabolic Method**. This method utilizes a parabolic searching technique to find the minimum specific energy. This method is very fast, but it is only capable of finding a single minimum on the energy curve. A second method, **Multiple Critical Depth Search**, is capable of finding up to three minimums on the energy curve. If more than one minimum is found the program selects the answer with the lowest energy. Very often the program will find minimum energies at levee breaks and breaks due to ineffective flow settings. When this occurs, the program will not select these answers as valid critical depth solutions, unless there is no other answer available. The Multiple Critical Depth Search routine takes a lot of computation time. Since critical depth is calculated often, using this method will slow down the computations. This method should only be used when you feel the program is finding an incorrect answer for critical depth.

Split Flow Optimizations. This option allows the user to have the program optimize the split of flow at lateral structures, lateral diversions, stream junctions, and pump stations. When this option is selected, a window will appear as shown in Figure 7.9. As shown in Figure 7.9, there are four tabs to choose from. One tab is for each of the following: Junctions; Lateral Weirs/Diversions; Reach-Storage Areas; and Pump Stations.

When the **Lateral Weir/Diversion** tab is selected, a table with all of the lateral weirs/spillways and rating curves defined in the model will be displayed. To have the program optimize the split of flow between the main stream and a lateral weir/spillway (or rating curve), the user simply checks the column labeled "Optimize." If you do not want a particular lateral weir/spillway to be optimized, the user should not check the box. For the first iteration of the flow split optimization, the program assumes that zero flow is going out of the lateral structure. Once a profile is computed, the program will then compute flow over the lateral structure. The program then iteratively reduces the flow in the main channel, until a balance is reached between the main river and the lateral structure. The user has the option to enter an initial estimate of the flow going out the lateral structure. This can speed up the computations, and may allow the program to get to a solution that may not have otherwise been possible. This option is available by selecting "**Initial Split Flow Values**" from the "**Options**" menu of the Steady Flow Data editor.

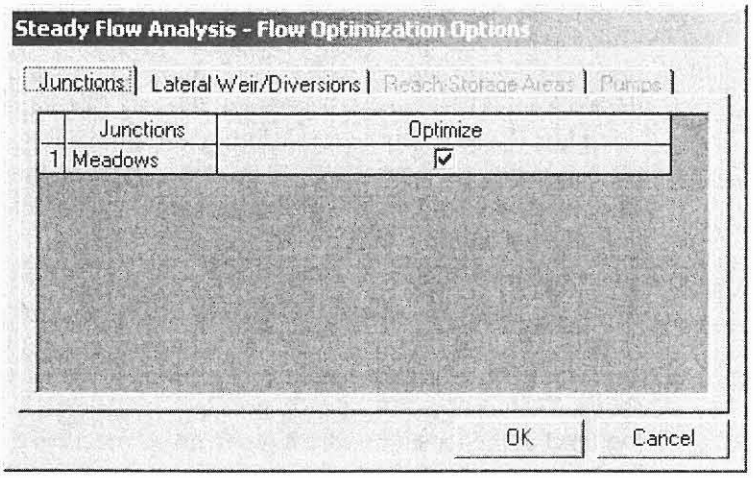


Figure 7.9 Split Flow Optimization Window

When the **Junction** tab is selected, the table will show all of the junctions in the model that have flow splits. To have the program optimize the split of flow at a junction, check the optimize column, otherwise leave it unchecked. Flow optimizations at junctions are performed by computing the water surface profiles for all of the reaches, then comparing the computed energy grade lines for the cross sections just downstream of the junction. If the energy in all the reaches below a junction is not within a specified tolerance (0.02 feet), then the flow going to each reach is redistributed and the profiles

are recalculated. This methodology continues until a balance is reached.

When the **Reach – Storage Areas** tab is selected, a window will appear displaying all of the storage areas that are upstream boundaries to river reaches. If optimization is set on the program will automatically compute the stage in the storage area based in the flow in the river reach.

The final tab is for **Pumps**. When this tab is pressed a table will appear showing all of the locations where pump stations are connected to the main rivers. The user can then turn on optimization for the split of flow between the main river and the pump station.

Check Data Before Execution. This option provides for comprehensive data input checking. When this option is turned on, data input checking will be performed when the user presses the compute button. If all of the data are complete, then the program allows the steady flow computations to proceed. If the data are not complete, or some other problem is detected, the program will not perform the steady flow analysis, and a list of all the problems in the data will be displayed on the screen. If this option is turned off, data checking is not performed before the steady flow execution. The default is that the data checking is turned on.

Set Log File Output Level. This option allows the user to set the level of the Log file. The Log file is a file that is created by the computational program. This file contains information tracing the program process. Log levels can range between 0 and 10, with 0 resulting in no Log output and 10 resulting in the maximum Log output. In general, the Log file output level should not be set unless the user gets an error during the computations. If an error occurs in the computations, set the log file level to an appropriate value. Re-run the computations and then review the log output, try to determine why the program got an error.

When the user selects **Set Log File Output Level**, a window will appear as shown in Figure 7.10. The user can set a "Global Log Level," which will be used for all cross sections and every profile. The user can also set log levels at specific locations for specific profiles. In general, it is better to only set the log level at the locations where problems are occurring in the computations. To set the specific location log level, first select the desired reach and river station. Next select the log level and the profile number (the log level can be turned on for all profiles). Once you have everything set, press the **Set** button and the log level will show up in the window below. Log levels can be set at several locations individually. Once all of the Log Levels are set, press the **OK** button to close the window.

Warning !!! - Setting the log output level to 4 or 5 can result in very large log file output. Log level values of 6 or larger can result in extremely large log files.

View Log File. This option allows the user to view the contents of the log file. The interface uses the Windows Write program to accomplish this. It is

up to the user to set an appropriate font in the Write program. If the user sets a font that uses proportional spacing, the information in the log file will not line up correctly. Some fonts that work well are: Line Printer; Courier (8 pt.); and Helvetica (8 pt.). Consult your Windows user's manual for information on how to use the Write program.

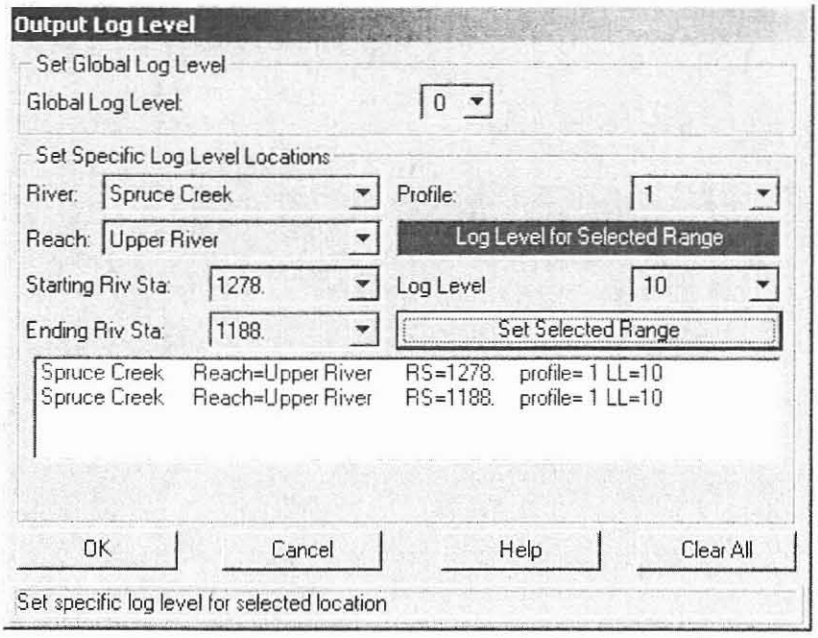


Figure 7.10 Log File Output Level Window

Starting the Computations

Once all of the data have been entered, and a Plan has been defined, the steady flow computations can be performed by pressing the **Compute** button at the bottom of the steady flow simulation window. Once the compute button is pressed, a separate window will appear showing you the progress of the computations. The information that appears in the window is only there as an indicator of the programs progress during the computations. This window is not intended to be used for viewing any output. When the computations have been completed, the user can close the computations window by clicking the close button at the bottom of the window. If the computations ended with a message stating "**Finished Steady Flow Simulation**," the user can then begin to review the output.

CHAPTER 8

Performing an Unsteady Flow Analysis

This chapter shows how to calculate unsteady flow water surface profiles. The chapter is divided into two parts. The first part explains how to enter unsteady flow data and boundary conditions. The second part describes how to develop a plan and perform the calculations.

Contents

- Entering and Editing Unsteady Flow Data
- Performing Unsteady Flow Calculations
- Calibration of Unsteady Flow Models
- Model Accuracy, Stability, and Sensitivity

Entering and Editing Unsteady Flow Data

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 in Figure 8.1.

Unsteady Flow Data

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

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

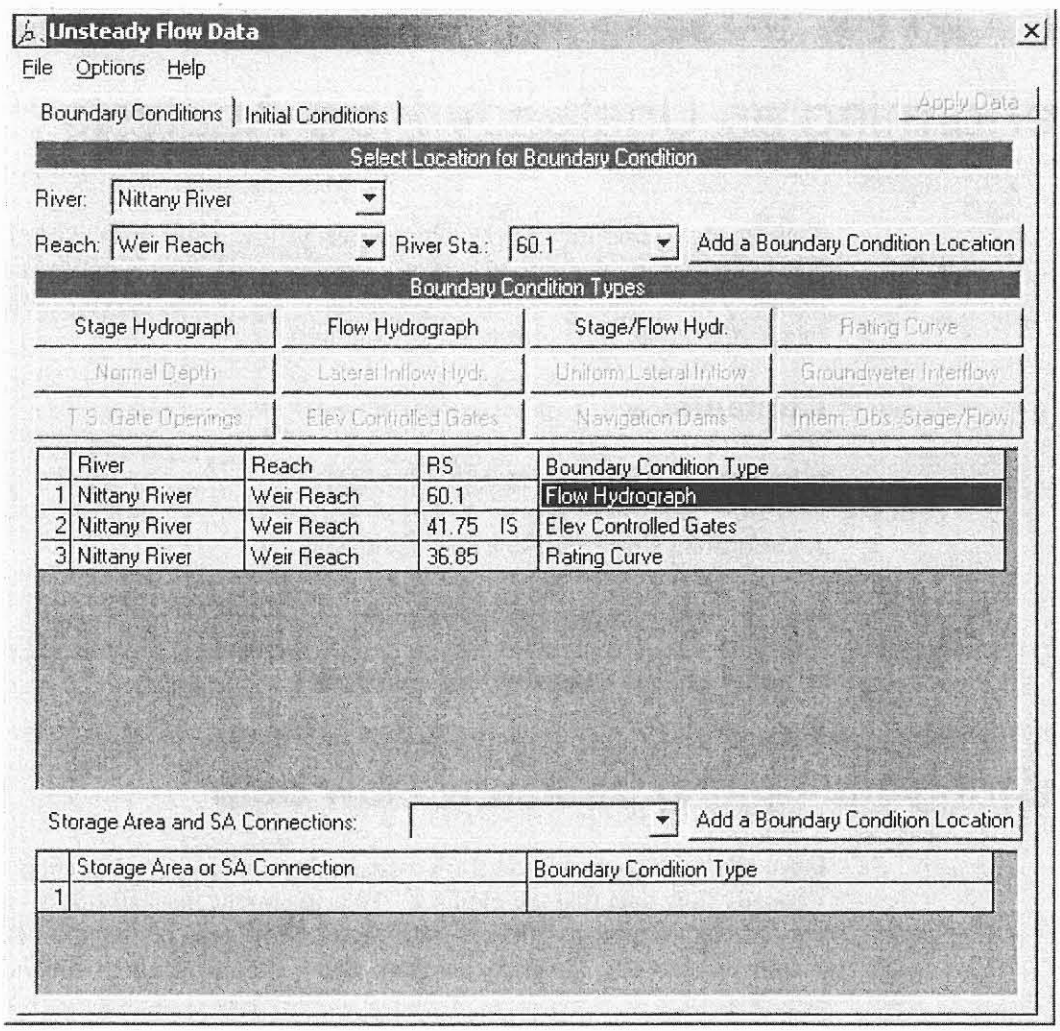


Figure 8.1 Unsteady Flow Data Editor

Boundary Conditions

There are several different types of boundary conditions available to the user. The following is a short discussion of each type:

Flow Hydrograph:

A flow hydrograph can be used as either an upstream boundary or downstream boundary condition, but is most commonly used as an upstream boundary condition. When the flow hydrograph button is pressed, the window shown in Figure 8.2 will appear. As shown, the user can either read the data from a HEC-DSS (HEC Data Storage System) file, or they can enter the hydrograph ordinates into a table. If the user selects the option to read the data from DSS, they must press the "**Select DSS File and Path**" button. When this button is pressed a DSS file and pathname selection screen will

appear as shown in Figure 8.3. 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.

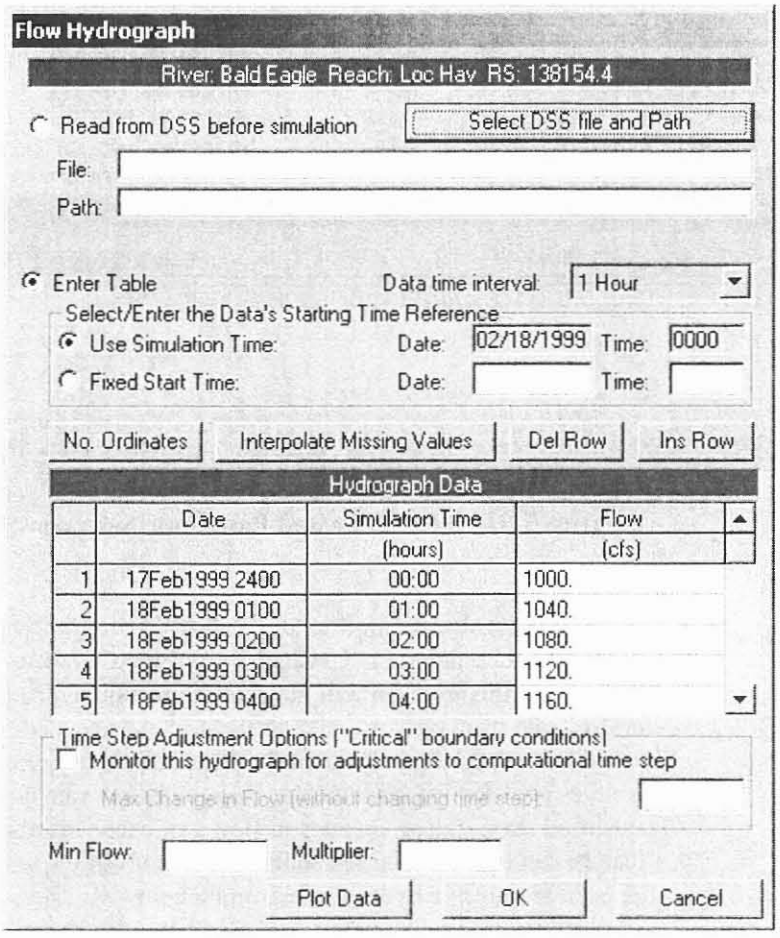


Figure 8.2 Example Flow Hydrograph Boundary Condition

The user also has the option of entering a flow hydrograph directly into a table, as shown in Figure 8.2. 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 hydrograph that they enter will always start at the beginning of the simulation time window. The simulation starting date and time is shown next to this box, but is grayed out. If the user selects "Fixed Start Time" then the hydrograph is entered starting at a user specified time and date. Once a starting date and time is selected, the user can

then begin entering the data.

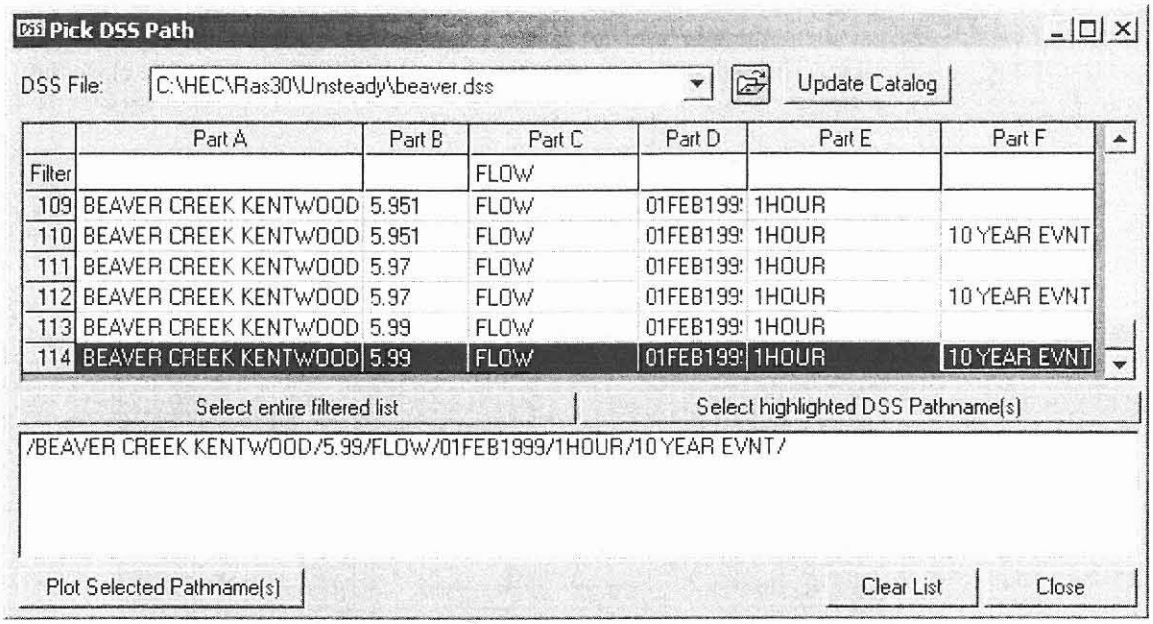


Figure 8.3 HEC-DSS File and Pathname Selection Screen

An option listed at the bottom of 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 max. Large changes in flow can cause instabilities. The use of this feature can help to keep the solution of the program stable. This feature can be used at multiple hydrographs simultaneously. The software will evaluate all of the hydrographs, then calculate a time slice based on the hydrograph with the largest percentage increase over the user specified maximum flow change.

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 stability 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.

Stage Hydrograph:

A stage hydrograph can be used as either an upstream or downstream boundary condition. The editor for a stage hydrograph is similar to the flow hydrograph editor (Figure 8.2). The user has the choice of either attaching a HEC-DSS file and pathname or entering the data directly into a table.

Stage and Flow Hydrograph:

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.

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. Shown in Figure 8.4 is the editor with data entered into the table. The downstream rating curve is a single valued relationship, and does not reflect a loop in the rating, which may occur during an event. This assumption may cause errors in the vicinity of the rating curve. The errors become a problem for streams with mild gradients where the slope of the water surface is not steep enough to dampen the errors over a relatively short distance. When using a rating curve, make sure that the rating curve is a sufficient distance downstream of the study area, such that any errors introduced by the rating curve do not affect the study reach.

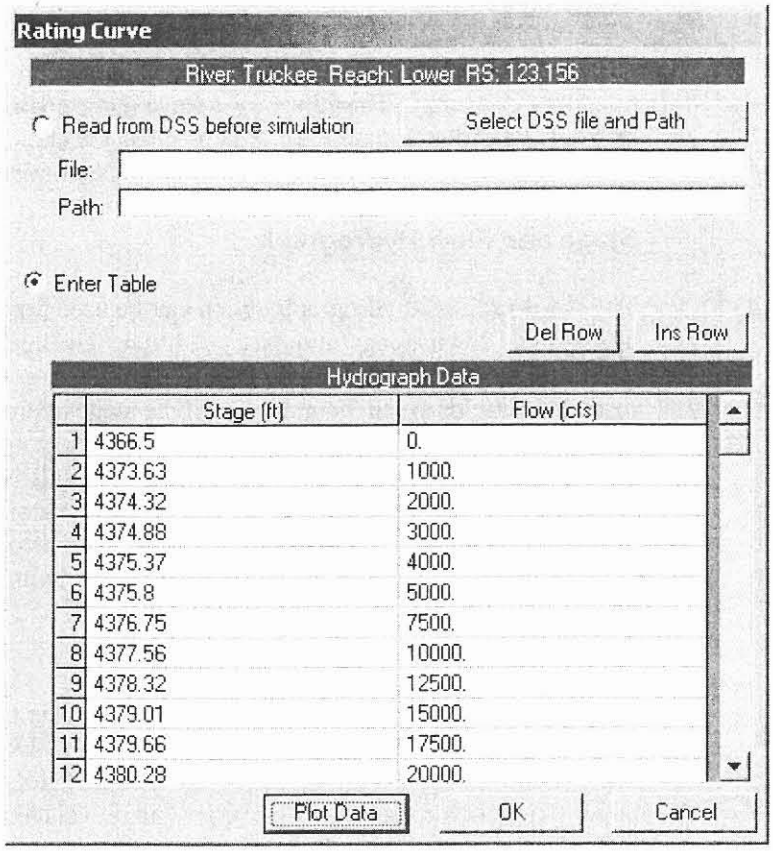


Figure 8.4 Example Rating Curve Boundary Condition Editor

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 it should placed far enough downstream, such that any errors it produces will not affect the results at the study reach.

Lateral Inflow Hydrograph:

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

Uniform Lateral Inflow Hydrograph:

The Uniform Lateral Inflow Hydrograph is used as an internal boundary condition. This option 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 in from DSS, or entered by hand into a table.

Groundwater Interflow:

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

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 gated spillway connecting two storage areas. The user has the option of reading the data from a DSS file or entering the data into a table from within the editor. Figure 8.5 shows an example of the Times Series of Gate Openings editor. As shown in Figure 8.5, the user first selects a gate group, then either attaches a DSS pathname to that group or enters the data into the table. This is done for each of the gate groups contained within the particular hydraulic structure.

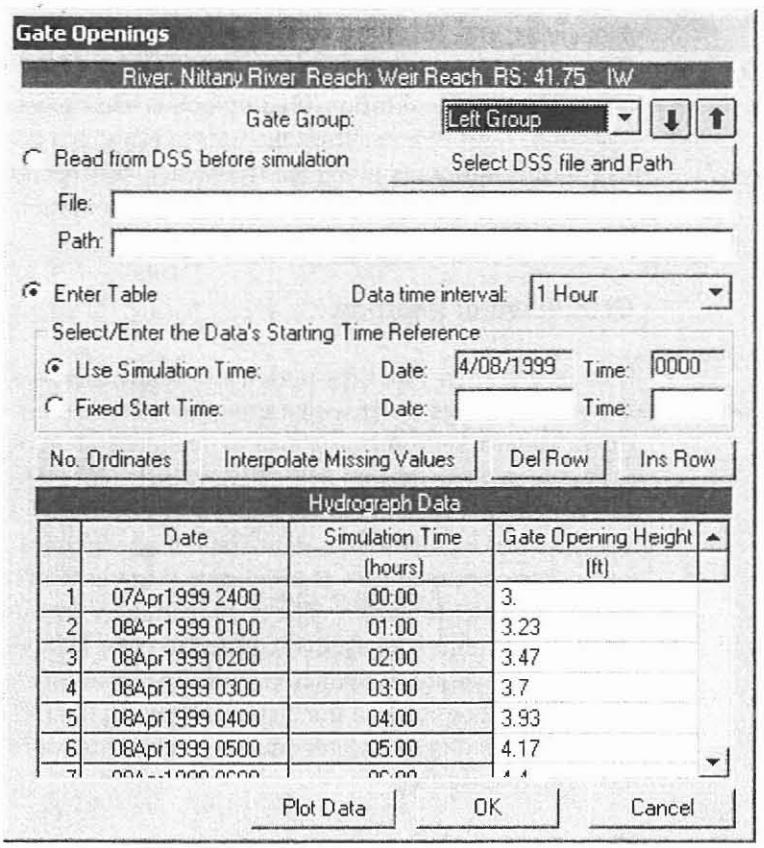


Figure 8.5 Example Time Series of Gate Openings Editor

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

Elevation Controlled Gate:

This option allows the user to control the opening and closing of gates based on the elevation of the water surface upstream of the structure. A gate begins to open when a user specified elevation is exceeded. The gate opens at a rate specified by the user. As the water surface goes down, the gate will begin to close at a user specified elevation. The closing of the gate is at a user specified rate (feet/min.). The user must also enter a maximum and minimum gate opening, as well as the initial gate opening. Figure 8.6 shows and example of this editor.

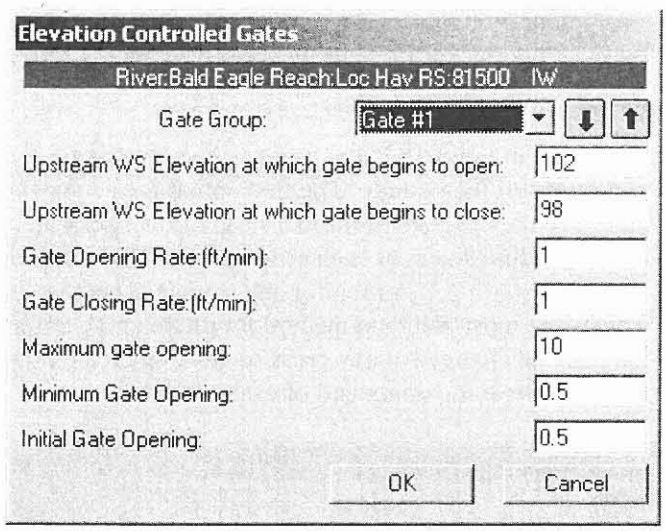


Figure 8.6 Elevation Controlled Gate Editor

Navigation Dam:

This option allows the user to define an inline gated structure as a hinge pool operated navigation dam. The user specifies stage and flow monitoring locations, as well as a range of stages and flow factors. This data is used by the software to make decisions about gate operations in order to maintain water surface elevations at the monitor locations. A detailed discussion about Navigation Dams can be found in Chapter 16 of the user's manual.

Internal Observed Stage and Flow Hydrograph:

This option allows the user to enter an observed stage hydrograph or a stage and flow hydrograph, to be used as an internal boundary condition just upstream of an inline structure (inline weir/spillway, bridge, or culvert). If only an observed stage hydrograph is entered, the user must have values to cover the complete simulation range. If a stage and flow hydrpograph is entered, the stage hydrograph is used as the observed boundary until the stage hydrograph runs out of data; afterward the flow hydrograph is used. The end of data in the stage hydrograph is identified by the HEC-DSS missing data code, -901.0. The stage and flow hydrographs can either be entered into a table or from HEC-DSS. The mixed boundary condition is primarily used for forecast models where the stage data is observed up to the forecast time and the flow hydrograph is the flow forecast.

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. Initial conditions are established from within the Unsteady Flow Data editor

by selecting the **Initial Conditions** tab. After the Initial Conditions tab is selected, the Unsteady Flow Data editor will appear as shown in Figure 8.7.

As shown in Figure 8.7, the user has two options for establishing the initial conditions of the system. 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 that are part of the system. This is the most common method for establishing initial conditions. Flow data can be changed at any cross section, but at a minimum the user must enter a flow at the upper end of each reach.

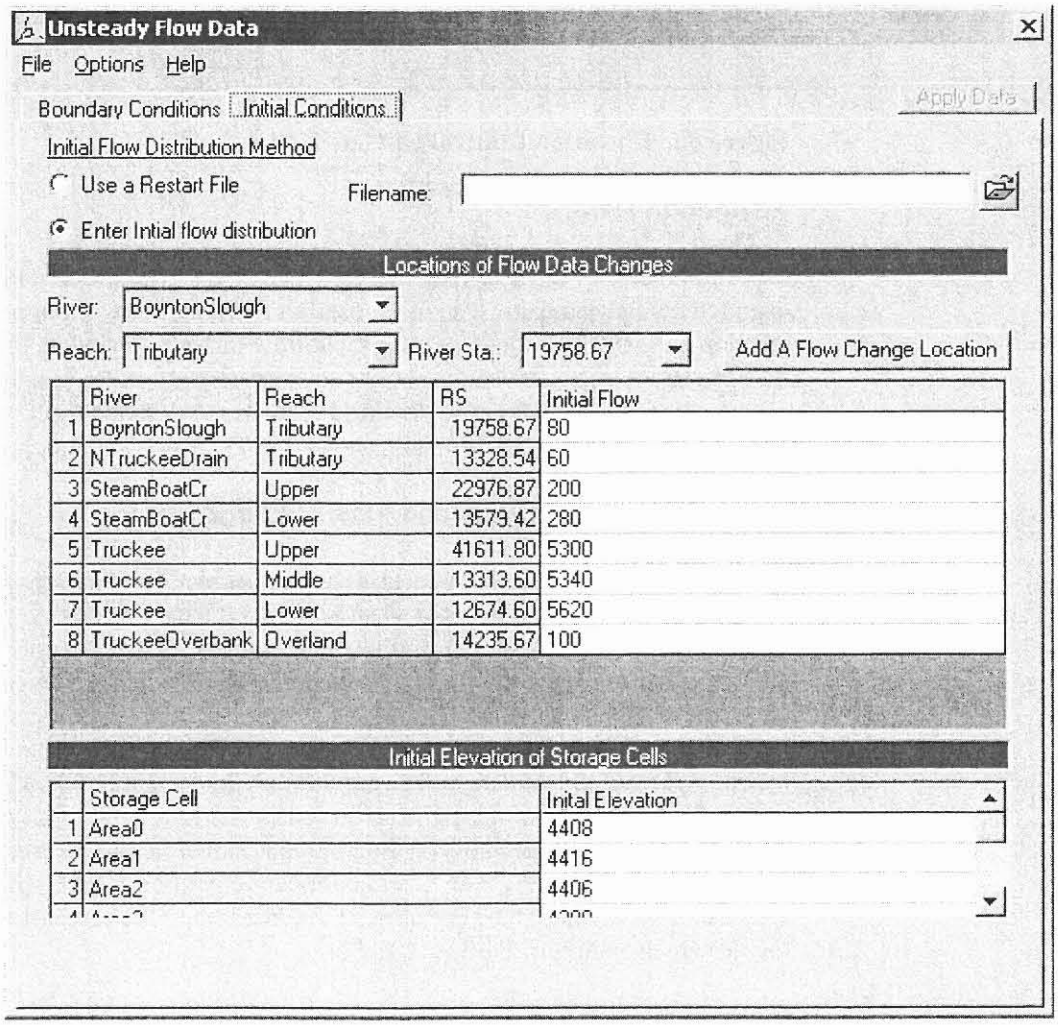


Figure 8.7 Initial Conditions Editor

A second method is to read in a file of stages and flows that were written from a previous run, which is called a "Restart File." This option is often used when running a long simulation time that must be divided into shorter periods. The output from the first period is used as the initial conditions for the next period, and so on. Additionally, this option may be used when the

software is having stability problems at the very beginning of a run. Occasionally the model may go unstable at the beginning of a simulation because of bad initial conditions. When this happens, one way to fix the problem is to run the model with all the inflow hydrographs set to a constant flow, and set the downstream boundaries to a high tailwater condition. Then run the model and decrease the tailwater down to a normal stage over time (use a stage hydrograph downstream boundary to do this). Once the tailwater is decreased to a reasonable value, those conditions can be written out to a file, and then used as the starting conditions for the unsteady flow run.

Unsteady Flow Data Options

Several options are available from the Unsteady Flow Data editor to assist users in entering and viewing the data. These features can be found under the **Options** menu at the top of the window. The following options are available:

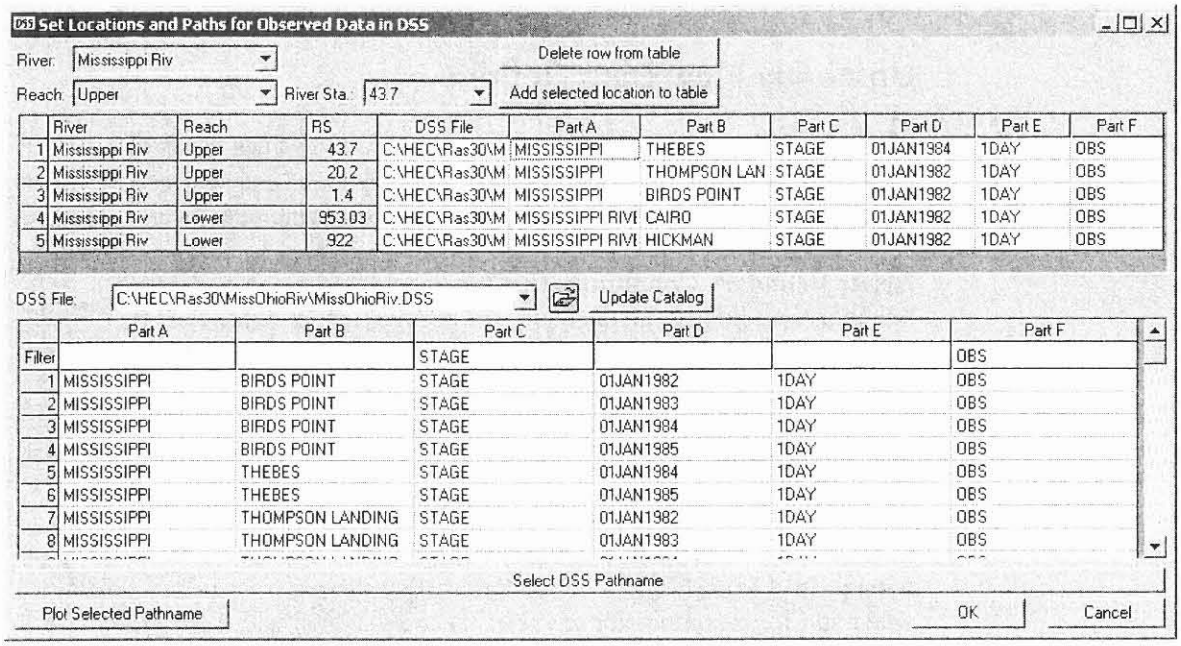
Delete Boundary Condition. This option allows the user to delete a boundary condition from the table. To use this option, first select the row to be deleted with the mouse pointer. Then select **Delete Boundary Condition** from the options menu. The row will be deleted and all rows below it will move up one. Only user inserted boundary conditions can be deleted from the table. If the boundary condition is an open end of the system, the system will not allow that boundary to be deleted. There must always be some type of boundary condition at all the open ends of the system.

Internal RS Initial Stages. This option allows the user to specify starting water surface elevations for any internal cross section within the system. A common application of this would be to specify the starting pool elevation for the first cross section upstream of a dam (modeled with the inline weir/spillway option). The user specifies locations and water surface elevations, which are then used to establish the initial conditions of the system at the beginning of a run.

Observed Data In DSS. This option allows the user to attach observed data pathnames from a HEC-DSS file to specific river stations within the model. When an observed data pathname is attached to a specific river station location, the user can get a plot of the observed flow or stage hydrograph on the same plot as the computed flow and stage hydrographs. Additionally the observed data will show up on profile and cross section plots.

To use this option, the user selects **Observed Data In DSS** from the **Options** menu of the Unsteady Flow Data editor. When this option is selected a window will appear as shown in Figure 8.8. As shown in the figure below, the user first selects a river, reach, and river station. Then the user presses the **Add selected location to table** button in order to select a location to attach observed data. This should be done for all the locations in which you have observed data. The next step is to open up the DSS file that contains the observed data. This is accomplished by pressing the open file button, which is next to the DSS filename field. When a DSS file is selected, a list of the available pathnames contained in that DSS file will show up in the lower

table. To attach a DSS pathname to a particular river station, first select the river station row from the upper table. Then select the DSS pathname row from the lower table. Finally, press the button labeled **Select DSS Pathname**. Repeat this process for every location in which you wish to attach observed data. If you are going to have more than one data type (such as stage and flow) at a particular river station, you must have two entries in the upper table for that river station.



	River	Reach	RS	DSS File	Part A	Part B	Part C	Part D	Part E	Part F
1	Mississippi Riv	Upper	43.7	C:\HEC\Ras30\M	MISSISSIPPI	THEBES	STAGE	01JAN1984	1DAY	OBS
2	Mississippi Riv	Upper	20.2	C:\HEC\Ras30\M	MISSISSIPPI	THOMPSON LAN	STAGE	01JAN1982	1DAY	OBS
3	Mississippi Riv	Upper	1.4	C:\HEC\Ras30\M	MISSISSIPPI	BIRDS POINT	STAGE	01JAN1982	1DAY	OBS
4	Mississippi Riv	Lower	953.03	C:\HEC\Ras30\M	MISSISSIPPI RIVI	CAIRO	STAGE	01JAN1982	1DAY	OBS
5	Mississippi Riv	Lower	922	C:\HEC\Ras30\M	MISSISSIPPI RIVI	HICKMAN	STAGE	01JAN1982	1DAY	OBS

	Part A	Part B	Part C	Part D	Part E	Part F
Filter			STAGE			OBS
1	MISSISSIPPI	BIRDS POINT	STAGE	01JAN1982	1DAY	OBS
2	MISSISSIPPI	BIRDS POINT	STAGE	01JAN1983	1DAY	OBS
3	MISSISSIPPI	BIRDS POINT	STAGE	01JAN1984	1DAY	OBS
4	MISSISSIPPI	BIRDS POINT	STAGE	01JAN1985	1DAY	OBS
5	MISSISSIPPI	THEBES	STAGE	01JAN1984	1DAY	OBS
6	MISSISSIPPI	THEBES	STAGE	01JAN1985	1DAY	OBS
7	MISSISSIPPI	THOMPSON LANDING	STAGE	01JAN1982	1DAY	OBS
8	MISSISSIPPI	THOMPSON LANDING	STAGE	01JAN1983	1DAY	OBS

Figure 8.8 Editor for Establishing Locations of Observed Data

Minimum Flow and Flow Ratio Table. This option brings up a global editor that will show all the locations in which flow hydrographs have been attached as boundary conditions. The editor allows the user to enter a minimum flow or a flow factor for each flow hydrograph boundary condition. The minimum flow option will prevent any flow read from either HEC-DSS or a user entered hydrograph from going lower than the user specified minimum. Values that are lower than the minimum specified are automatically changed to the minimum value. The flow factor option allows the user to specify a factor to be multiplied by all ordinates of the flow hydrograph. This option is commonly used in planning type studies for performing sensitivity analysis (i.e. what if the flow were 20% higher?).

Saving The Unsteady Flow Data

The last step in developing the unsteady flow data is to save the information to a file. To save the data, select the **Save Unsteady Flow Data As** from the **File** menu on the Unsteady Flow Data editor. A pop-up window will appear prompting you to enter a title for the data.

Performing Unsteady Flow Calculations

Once all of the geometry and unsteady flow data have been entered, the user can begin performing the unsteady flow calculations. To run the simulation, go to the HEC-RAS main window and select **Unsteady Flow Analysis** from the **Run** menu. The Unsteady Flow Analysis window will appear as in Figure 8.9 (except yours may not have a Plan title and short ID).

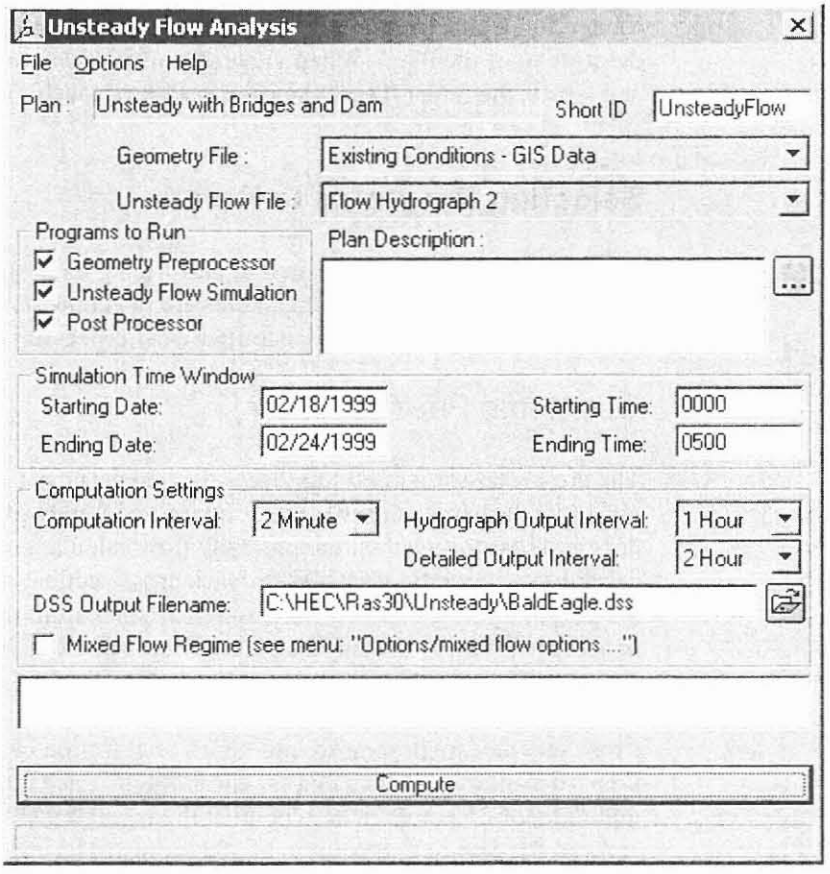


Figure 8.9 Unsteady Flow Analysis Window

Defining A Plan

The first step in performing a simulation is to put together a Plan. The Plan defines which geometry and unsteady flow data are to be used, as well as provides a description and short identifier for the run. Also included in the Plan information are the selected programs to be run; simulation time window; computation settings; and the simulation options.

Before a Plan is defined, the user should select which geometry and unsteady flow data will be used in the Plan. To select a geometry or unsteady flow file, press the down arrow button next to the desired data type. When this button is pressed, a list will appear displaying all of the available files of that type

that are currently available for the project. Select the geometry and unsteady flow file that you want to use for the current Plan.

To establish a Plan, select **Save Plan As** from the **File** menu on the Unsteady Flow Analysis window. When **Save Plan As** is selected, a window will appear prompting you to enter a title for the Plan. After you enter the title, press the **OK** button to close the window and accept the title. The user will also be prompted to enter a short identifier for the Plan. The short identifier is limited to 12 characters. It is very important to enter a short identifier that is descriptive of the Plan. When viewing multiple plan output from the graphics and tables, the Short ID will be used to identify each Plan.

Selecting Programs to Run

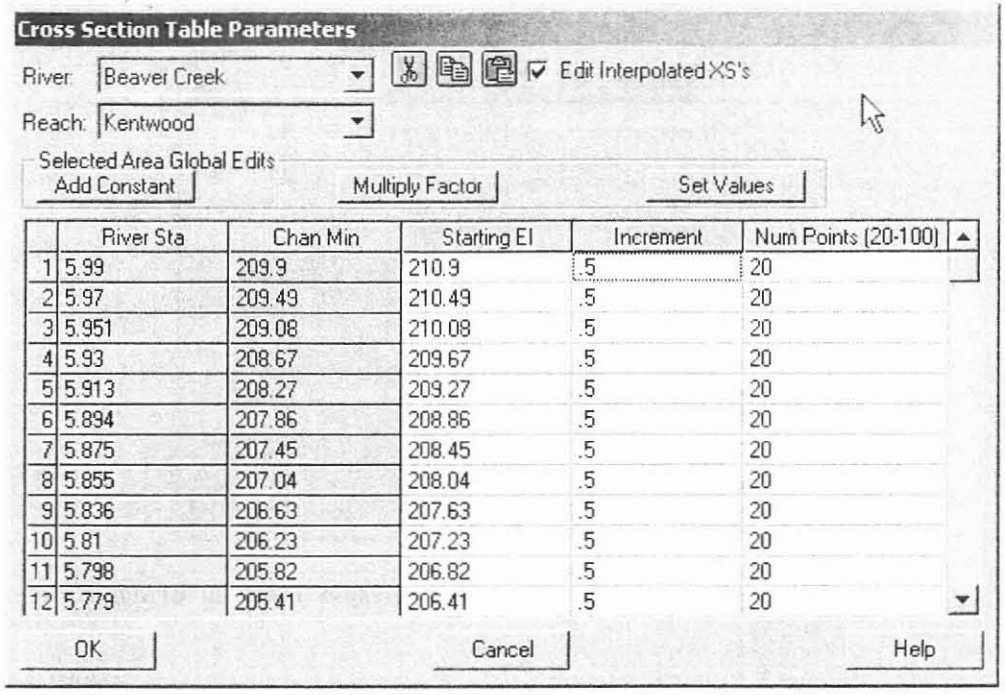
There are three components used in performing an unsteady flow analysis within HEC-RAS. These components are: a geometric data pre-processor; the unsteady flow simulator; and an output post-processor.

Geometric Pre-Processor

The pre-processor is used to process the geometric data into a series of hydraulic properties tables, rating curves, and family 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.**

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), and can have up to a maximum of 100 points. 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 to large, the tables will not have enough detail to accurately depict changes in area, conveyance, and storage with respect to elevation.

The interval for the cross section tables is defined as part of the geometric data. To set this interval, the user selects the **HTab Parameters** (**H**ydraulic **Tab**le Parameters) button from the Geometric Data editor. When this option is selected, a window will appear as shown in Figure 8.10.



	River Sta	Chan Min	Starting El	Increment	Num Points (20-100)
1	5.99	209.9	210.9	.5	20
2	5.97	209.49	210.49	.5	20
3	5.951	209.08	210.08	.5	20
4	5.93	208.67	209.67	.5	20
5	5.913	208.27	209.27	.5	20
6	5.894	207.86	208.86	.5	20
7	5.875	207.45	208.45	.5	20
8	5.855	207.04	208.04	.5	20
9	5.836	206.63	207.63	.5	20
10	5.81	206.23	207.23	.5	20
11	5.798	205.82	206.82	.5	20
12	5.779	205.41	206.41	.5	20

Figure 8.10 Hydraulic Table Parameters for Cross Sections

As shown in Figure 8.10, the table contains three columns in which the user can enter a 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.

Hydraulic structures, such as bridges and culverts, are converted into families of rating curves that describe the structure as a function of tailwater, flow, and headwater. The user can set several parameters that can be used in defining the curves. To set the parameters for the family of rating curves, the user can select the **"HTab Parameters"** button from the Bridge and Culvert editor or from the Hydraulic connection editor. When this button is pressed, the window in Figure 8.11 will appear:

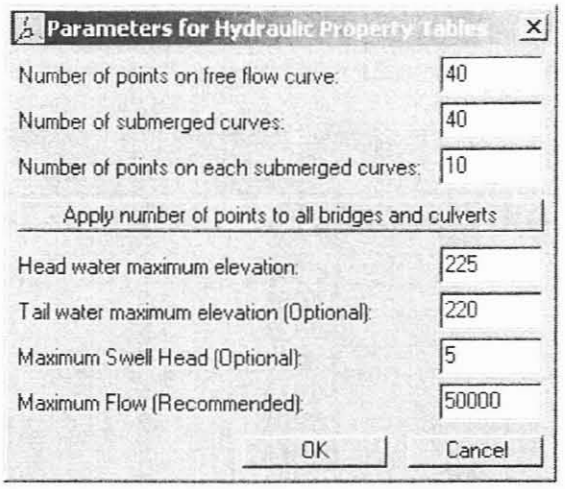


Figure 8.11 Hydraulic Properties Table for Bridges/Culverts

As shown in Figure 8.11, the user can set the number of points to be computed on the free-flow rating curve (maximum of 50 points); the number of submerged curves to be computed (maximum of 50); and the number of points on the submerged curves (maximum of 20). The default values for these parameters are 40, 40, and 10 respectively. Additionally, the user can refine the curves by setting limits on the extent of the curves. This can be accomplished by entering the head water maximum elevation (reguired), tail water maximum elevation, maximum swell head (difference between the head water and tailwater), and the maximum possible flow (recommended). **In general, the curves will come out better if the user enters a headwater maximum elevation and a maximum flow rate.**

Structures that are gated, such as gated spillways, are not converted into curves because it would require a new family of curves for each possible gate setting. The hydraulics through gated structures is calculated on the fly during the unsteady flow calculations. No hydraulic table parameters are required for gated structures.

Unsteady Flow Simulation

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) and modified by HEC. The unsteady flow simulation is actually a three-step process. First a program called RDSS (Read DSS data) runs. This software reads data from a HEC-DSS file and then converts all of the boundary condition time series data into the user specified computation interval. Next, the UNET program runs. This software reads the hydraulic properties tables computed by the pre-processor, as well as the boundary conditions and flow data from the interface and the RDSS program. The program then performs the unsteady flow calculations. The final step is a program called TABLE. This software takes the results from the UNET unsteady flow run and writes them to a HEC-DSS file.

Post-Processor

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 unsteady flow computations only compute stage and flow at all of the computation nodes, as well as 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.

By default, the Post-Processor will compute detailed output for a maximum stage water surface profile. This profile does not represent any specific instance in time, but rather represents a profile of the maximum stage that occurred at each cross section during the entire simulation. This profile is often useful for getting a quick view of the maximum extent of flooding during a specific event.

In addition to the maximum water surface profile, the user can request the software to write out a series of instantaneous profiles at a specific time interval. This is accomplished from the **Computation Settings** section of the **Unsteady Flow Analysis** window. The user turns on this option by selecting an interval from the box labeled **Detailed Output Interval.** The Post-Processor will then compute detailed output for each of the instantaneous profiles requested (Note: the Post-Processor is limited to 500 profiles). When the unsteady flow program runs, flow and stage water surface profiles are written to DSS for the entire system, starting with the beginning of the simulation and then at the user specified time interval for the entire simulation.

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.

At hydraulic structures such as bridges and culverts, the unsteady flow program only reports the stage just upstream and downstream of the structure. During the Post-Processing of the results, the SNET program calculates the hydraulics of the structures by using the computed tailwater and flow, and then performing detailed hydraulic structure calculations. This is done so that the user can see detailed hydraulic information inside of the hydraulic structures for each of the profiles that are being post processed. However, this process can produce slightly different results for the upstream headwater elevation. Occasionally, you may notice a headwater elevation computed from the Post-Processor that is higher than the next upstream sections water surface. This difference is due to the fact that the unsteady flow simulation uses a pre-computed family of rating curves for the structure during the unsteady flow calculations. The program uses linear interpolation between the points of the rating curves to get the upstream headwater for a given flow and tailwater. The Post-Process performs the calculations through the structure and does not use rating curves (it solves the actual structure equations).

Once the Post-Processor is finished running, the user can view output from all of the HEC-RAS plots and tables. The maximum water surface profile and user specified instantaneous profiles can be viewed by selecting **Profiles** from the **Options** menu on each of the output windows (tables or plots). The overall maximum water surface profile will be labeled "**Max W.S.**", while the instantaneous profiles are labeled by the date and time. For example, a profile from January 5, 1999 at 1:00 p.m. would be labeled "**05Jan1999 1300**".

WARNING: Specifying a detailed output interval for post processing that is small can lead to long computational times and huge output files. Select this interval wisely, in that you only get detailed output when you really need it.

Simulation Time Window

The user is required to enter a time window that defines the start and end of the simulation period. The time window requires a starting date and time and an ending date and time. The date must have a four digit year and can be entered in either of the two following formats: **05Jan2000** or **01/05/2000**. The time field is entered in military style format (i.e. 1 p.m. is entered as 1300).

Computation Settings

The Computation Settings area of the Unsteady Flow Analysis window contains: the computational interval; hydrograph output interval; detailed output interval; the name and path of the output DSS file, and whether or not the program is run in a mixed flow regime mode.

The **computation interval** is used in the unsteady flow calculations. This is probably one of the most important parameters entered into the model. Choosing this value should be done with care and consideration as to how it will affect the simulation. The computation interval should be based on several factors. First, the interval should be small enough to accurately describe the rise and fall of the hydrographs being routed. A general rule of thumb is to use a computation interval that is equal to or less than the time of rise of the hydrograph divided by 24. In other words, if the flood wave goes from its base flow to its peak flow in 24 hours, then the computation interval should be equal to or less than 1 hour.

Additional considerations must be made for hydraulic structures, such as bridges, culverts, weirs, and gated spillways. Within bridges and culverts, when the flow transitions from unsubmerged to submerged flow, the water surface upstream of the structure can rise abruptly. This quick change in water surface elevation can cause the solution of the unsteady flow equations to go unstable. One solution to this problem is to use a very small time step, on the order of 1 to 5 minutes. This allows the module to handle the changes in stage in a more gradual manner. Additionally, when gates are opened or when flow just begins to go over a lateral weir, the change in stage and flow can be dramatic. Again, these types of quick changes in stage and flow can cause the solution of the unsteady flow equations to go unstable. The only solution to this problem is to shorten the computational time step to a very short interval. This may require the user to set the value as low as 1 to 5 minutes. The time step should be adjusted to find the largest value that will still solve the equations accurately. Additional variables that affect stability are the number of iterations and the Theta weighting factor. These two variables are discussed under the calculation tolerances section below.

The **Hydrograph Output Interval** is used to define at what interval the computed stage and flow hydrographs will be written to HEC-DSS. This interval should be selected to give an adequate number of points to define the shape of the computed hydrographs without losing information about the peak or volume of the hydrographs. This interval must be equal to or larger than the selected computation interval.

The **Detailed Output Interval** field allows the user to write out profiles of water surface elevation and flow at a user specified interval during the simulation. Profiles are not written for every computational time step because it would require too much space to store all of the information for most jobs. Also, when the Post-Processor is run, the program will compute detailed hydraulic information for each one of the instantaneous profiles that are written. This option is turned on by selecting an interval from the drop-down

box next to the detailed hydrograph output label. The selected interval must be equal to or greater than the computation interval. However, it is suggested that you make this interval fairly large, in order to reduce the amount of post-processing and storage required for a detailed hydraulic output. One example for selecting this variable would be, if the time window of the simulation was set at 72 hours, then one might want to set the instantaneous profiles to an interval of every 6 hours. This would equate to 13 profiles being written out and having detailed hydraulic information computed for them.

The field labeled **DSS Output Filename** is required before an execution can be made. The program will always write some results to a HEC-DSS file, so the user is required to select a path and filename to be used for this information.

Mixed Flow Regime. When this option is selected, the program will run in a mode such that it will allow subcritical flow, supercritical flow, hydraulic jumps, and draw downs (sub to supercritical transitions). **This option should only be selected if you actually have a mixed flow regime situation.** The methodology used for mixed flow regime analysis is called the **Local Partial Inertia (LPI)** solution technique (Fread, 1996). When this option is turned on, the program monitors the Froude number at all cross section locations for each time step. As the Froude number gets close to 1.0, the program will automatically reduce the magnitude of the inertial terms in the momentum equation. Reducing the inertial terms increases the models stability. When the Froude number is equal to or greater than 1.0, the inertial terms are completely zeroed out and the model is essentially reduced to a diffusion wave routing procedure. For Froude numbers close to 1.0, the program will use partial inertial effects, and when the Froude number is low, the complete inertial effects are used.

Note: more information about mixed flow regime calculations can be found in Chapter 16 of the HEC-RAS User's manual.

Simulation Options

From the **Options** menu of the Unsteady Flow Analysis window, the following options are available: stage and flow output locations; flow distribution locations; flow roughness factors; seasonal roughness factors; calculation options and tolerances; output options; checking data before execution, and viewing the computation log.

Stage and Flow Output Locations. This option allows the user to specify locations where they want to have hydrographs computed and available for display. By default, the program sets locations of the first and last cross section of every reach. To set the locations, the user selects **Stage and Flow Output Locations** from the **Options** menu of the Unsteady Flow Analysis window. When this option is selected a window will appear as shown in Figure 8.12.
As shown in Figure 8.12, the user can select individual locations, groups of

cross sections, or entire reaches. Setting these locations is important, in that, after a simulation is performed, the user will only be able to view stage and flow hydrographs at the selected locations.

Flow Distribution Locations. This option allows the user to specify locations in which they would like the program to calculate flow distribution output. The flow distribution option allows the user to subdivide the left overbank, main channel, and right overbank, for the purpose of computing additional hydraulic information.

The user can specify to compute flow distribution information for all the cross sections (this is done by using the Global option) or at specific 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. The user can define up to 45 total slices. Each flow element (left overbank, main channel, and right overbank) must have at least one slice. The flow distribution output will be calculated for all profiles in the plan during the computations.

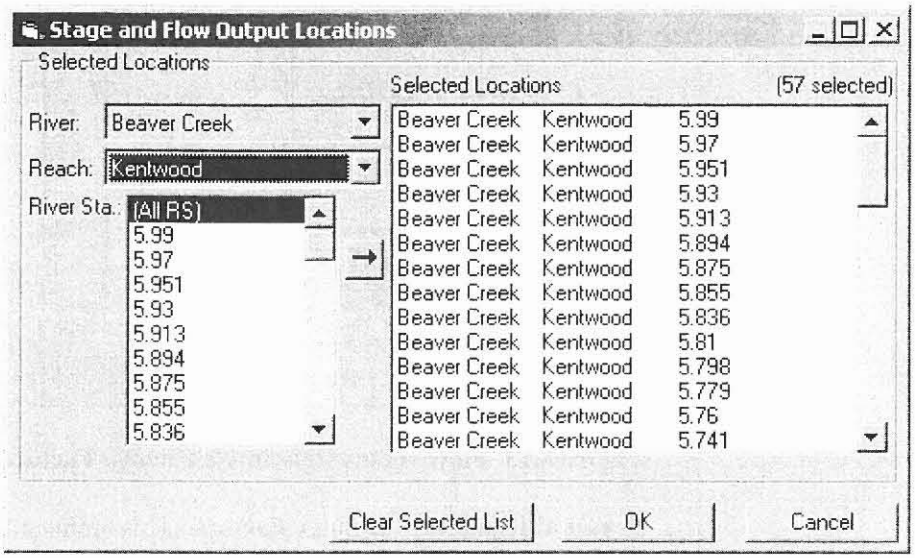


Figure 8.12 Stage and Flow Hydrograph Output Window

Flow Roughness Factors. This option allows the user to adjust roughness coefficients with changes in flow. This feature is very useful for calibrating an unsteady flow model for flows that range from low to high. Roughness generally decreases with increases flow and depth. This is especially true on larger river systems. This feature allows the user to adjust the roughness coefficients up or down in order to get a better match of observed data. To use this option, select **Flow Roughness Factors** from the **Options** menu of the Unsteady Flow Simulation manager. When this option is selected, a window will appear as shown in Figure 8.13.
As shown in Figure 8.13, the user first selects a river, reach, and a range of cross sections to apply the factors to. Next a starting flow, flow increment,

and a number of increments is entered. Finally, a roughness factor is entered into the table for each of the flows. The user can create several sets of these factors to cover a range of locations within the model. However, one set of factors cannot overlap with another set of factors. Hence, you can only apply one set of roughness change factors to any given cross section.

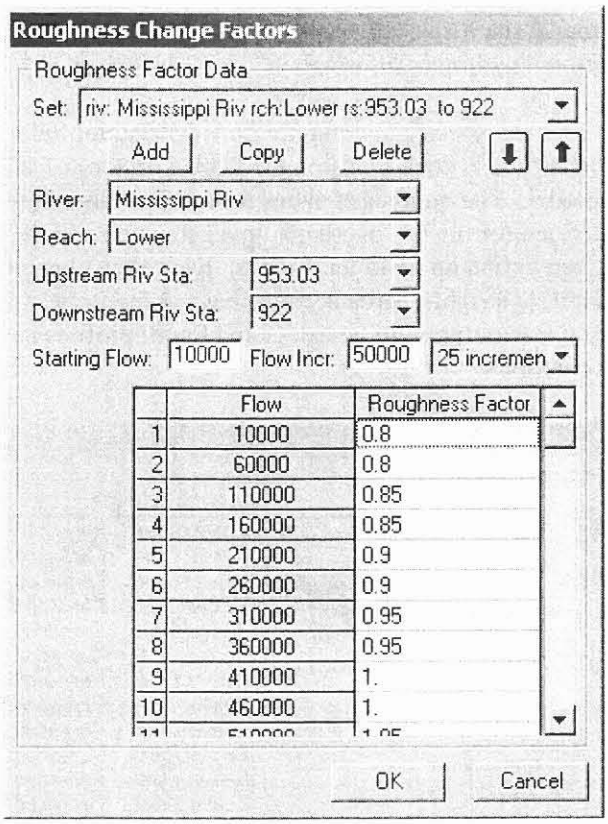


Figure 8.13 Flow versus Roughness Change Factors Editor

Seasonal Roughness Change Factors. This option allows the user to change roughness with time of year. This feature is most commonly used on larger river systems, in which temperature changes can cause changes in bed forms, which in turn causes changes in roughness. This factor can be applied in conjunction with the flow roughness change factors. When applying both, the seasonal roughness factor gets applied last.

To use this option, select **Seasonal Roughness Factors** from the **Options** menu of the Unsteady Flow Simulation manager. When this option is selected a window will appear as shown in Figure 8.14.

As shown in Figure 8.14, the user first selects a river, reach, and range of river station to apply the factors to. Next the user enters the day and month in the Day column, for each time that a new roughness factor will be entered. By default the program will automatically list the first of each month in this column. However, the user can change the day to whatever they would like. The final step is to then enter the roughness change factors.

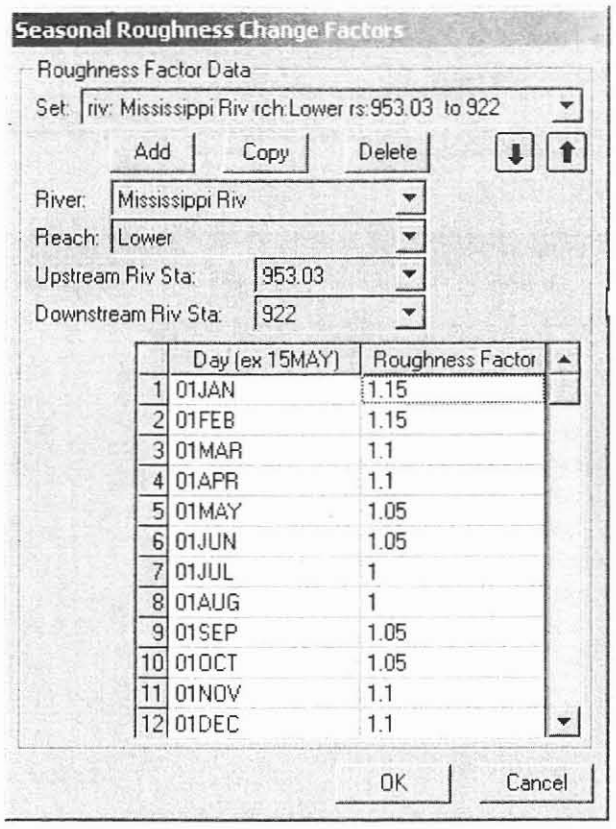


Figure 8.14 Seasonal Roughness Factors Editor

Unsteady Flow Encroachments. This option allows the user to perform an encroachment analysis using the unsteady flow simulation option. Currently, encroachments are limited to method 1 within the unsteady flow analysis module. In general the user should first perform the encroachment analysis with the steady flow computations module, as documented in Chapter 10 of this manual. Once a good steady flow encroachment analysis is completed, the final encroachments can be imported into the unsteady flow plan for further analysis and refinement. The user will need to have two unsteady flow plans, one without encroachments (representing the base flood) and one with encroachments (representing the encroached floodplain).

To add encroachments to an unsteady flow plan, the user selects **Unsteady Encroachments** from the **Options** menu of the Unsteady Flow Simulation editor. When this option is selected the following window will appear:

Unsteady Encroachments

River: (All Rivers)

Reach:

Get Encroachments from Steady Flow Plan ...

Selected Area Global Edits

Add Constant | Multiply Factor | Set Values

Unsteady Encroachments (Method 1 Style Encroachments)

	River	Reach	RS	Left Station	Right Station
1	Beaver Creek	Kentwood	5.99	67.53	1731.07
2	Beaver Creek	Kentwood	5.97	49.5	1763.97
3	Beaver Creek	Kentwood	5.951	36.23	1784.01
4	Beaver Creek	Kentwood	5.93	118.22	1607.87
5	Beaver Creek	Kentwood	5.913	153.61	1533.37
6	Beaver Creek	Kentwood	5.894	178.02	1479.7
7	Beaver Creek	Kentwood	5.875	189.82	1450.95
8	Beaver Creek	Kentwood	5.855	305.53	1296.01
9	Beaver Creek	Kentwood	5.836	202.89	1406.07
10	Beaver Creek	Kentwood	5.81	94.5	1564.15
11	Beaver Creek	Kentwood	5.798	93.36	1543.71
12	Beaver Creek	Kentwood	5.779	90.43	1523.64
13	Beaver Creek	Kentwood	5.76	79.29	1520.94
14	Beaver Creek	Kentwood	5.741	131.34	1481.37
15	Beaver Creek	Kentwood	5.72	180.16	1450.26

OK | Cancel

Figure 8.15 Unsteady Flow Encroachment Data Editor.

As shown in Figure 8.15, the user can enter a left station and a right station for the encroachments at each cross section. Additionally, the user has the option to import the encroachments calculated from a steady flow plan. This is accomplished by pressing the button labeled **Get Encroachments from Steady Flow Plan**, which is shown in the upper right part of the editor. When this button is pressed the user is asked to select a previously computed steady flow plan, and a specific profile from that plan. When the user presses the **OK** button, the program will go and get the final computed encroachments from that particular steady flow plan and profile.

Once all of the encroachments are entered, the user presses the **OK** button to have the interface accept the data. However, this information is not stored to the hard disk, the user must save the currently opened plan file for that to happen. The next step is to run the unsteady flow analysis with the encroachment data. The user should have two unsteady flow plans, one without encroachments and one with encroachments. Once both plans have been successfully executed, then comparisons between the plans can be made both graphically and in a tabular format.

Dam (Inline Structure) Breach. This option allows the user to perform a Dam Break analysis. The breach data is stored as "Plan" information. This is done so the user can try different breach locations, sizes, etc, without having to re-run the geometric pre-processor. Storing the data as Plan data is not that

important in a planning study, but it is very important when doing real time river forecasting. However, the user can get to the breach data in two different ways. First there is a button on the Inline Structure editor that is labeled **Breach** (Plan Data). Second, from the Unsteady Flow Simulation Manager, the user can select **Dam (Inline Weir) Breach** from the Options menu. When either option is selected, the following window will appear.

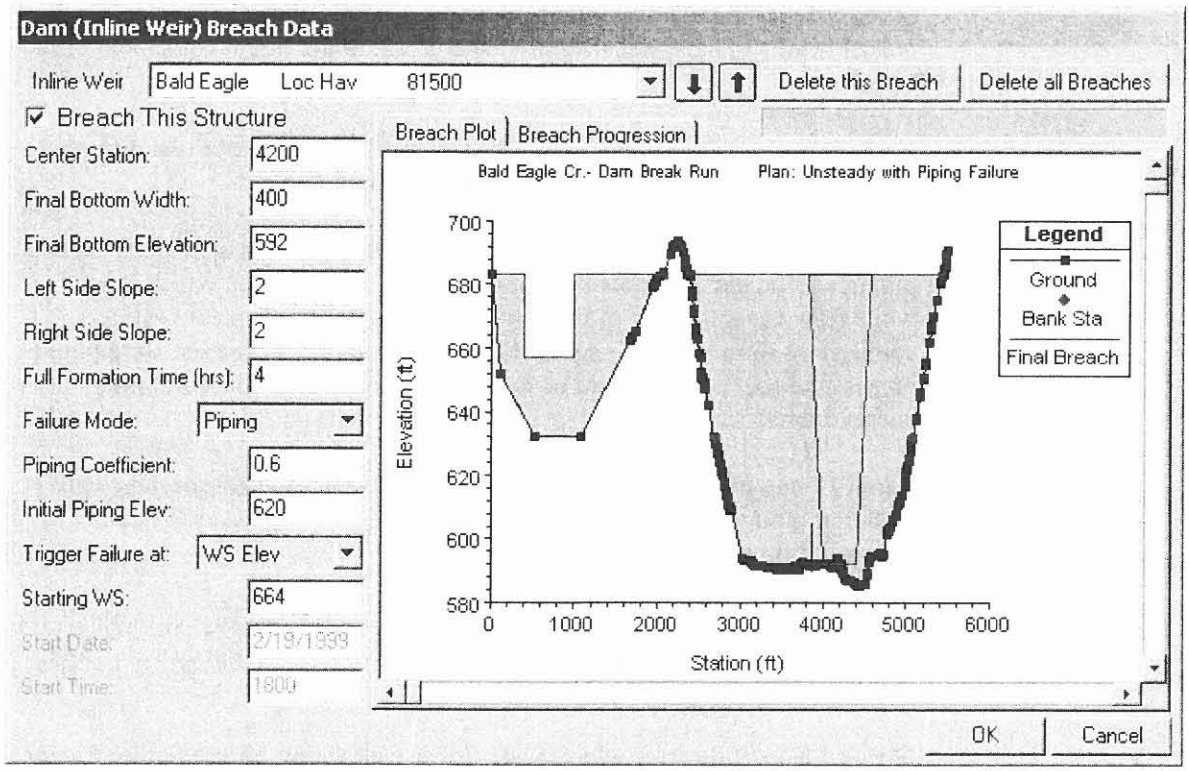


Figure 8.16 Dam Breach Editor.

As shown in Figure 8.16, the user selects a particular Inline Structure to perform the breach on. The following data must be entered for a breach:

Breach This Structure - This check box is used to decide if the program will perform the breach or not. In order for the breach to occur this box must be checked. This box was added to allow the user to turn certain breaches on or off, without loosing the user entered breach information.

Center Station - This field is used for entering the centerline stationing of the final breach.

Final Bottom Width - This field is used to enter the bottom width of the breach at its maximum size.

Final Bottom Elevation - This field is used to enter the elevation of the bottom of the breach after it has been fully developed.

Left Side Slope - This is the left side slope of the trapezoidal breach.

Right Side Slope - This is the right side slope of the trapezoidal breach.

Full Formation Time (hrs) - This field is used to enter the breach development time in hours. This time represents the duration from when the breach begins to have some significant erosion, to the full development of the breach.

Failure Mode - This option allows the user to choose between two different failure modes, an Overtopping failure and a Piping failure.

Piping Coefficient - If a piping failure mode is selected, the user must enter a piping coefficient. This coefficient is an Orifice coefficient, which is used while flow is coming out of the dam in a piping mode. Typical Orifice coefficients for a true designed orifice are around 0.8. However, for a piping breach, the coefficient should be lower to represent all of the additional energy losses occurring.

Initial Piping Elev. - If a piping failure mode is selected the user must enter an initial piping elevation. This elevation should be entered as the center of the piping flow while the breach develops.

Trigger Failure At - This field is used to select one of two trigger methods for initiating the breach. The two trigger methods are a water surface elevation or a specific time and date.

Starting WS - If the user selects water surface elevation for the failure trigger mode, then this field must be entered. This field represents the water surface elevation at which the breach should begin to occur.

Start Date - If the user selects a starting date and time as the failure trigger mode, then this field must be entered. This field is used to enter the date at which the breach will begin to occur.

Start Time - If the user selects a starting date and time as the failure trigger mode, then this field must be entered. This field is used to enter the time at which the breach will begin to occur.

In addition to all of the main breach information, the user also has the option to enter a user specified Breach Progression curve. By default the breach progression is assumed to be linear between the breach initiation and the full breach size (Full Formation Time). The user enters their breach progression curve by selecting the Breach Progression tab. When this tab is selected, the editor will now look like the following:

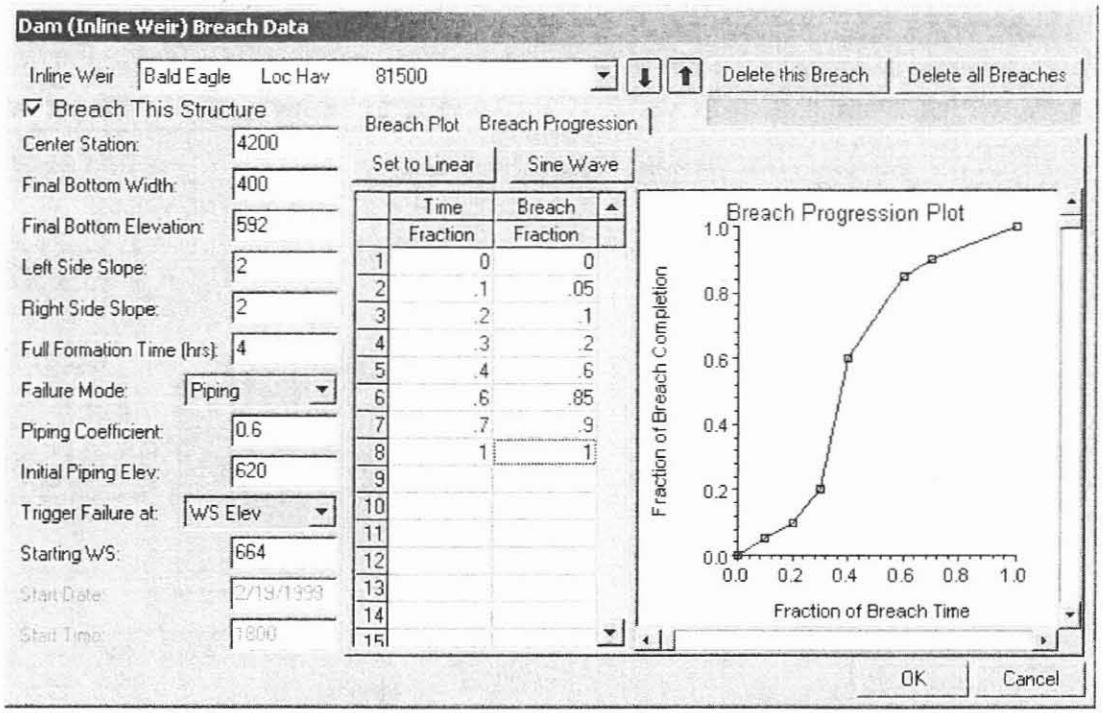


Figure 8.17 Dam Breach Editor with User Specified Breach Progression.

As shown in Figure 8.17, the user enters a Time Fraction (from zero to 1.0) and a Breach Fraction (from zero to 1.0). The user-entered data is plotted in the graphic next to the table. The breach progression curve is then used during the breach formation time to adjust the growth rate of the breach.

Note: More detailed information on performing a breach analysis can be found in the HEC-RAS User's Manual, chapter 16.

Levee (Lateral Structure) Breach. This option is very similar to the Dam Break option described previously. The only difference is that the breaching is performed on a levee. The options and data entered to describe the breach is the same as a Dam Break.

In order to use this option, the user must first define the levee as a lateral structure within HEC-RAS. The lateral weir profile is used to describe the top of the levee along the stream both at and between the cross sections. Second, a weir coefficient is entered for calculating the flow that may go overtop of the levee if the water surface gets high enough. Entering breach data for the levee can be accomplished from the lateral weir editor or from the **Levee (lateral structure) Breach** option from the Unsteady Flow Simulation window. The levee breaching data is stored as part of the unsteady flow plan file, just as it is for a dam break. When the levee breach option is selected, a breach editor will appear as shown in Figure 8.18.

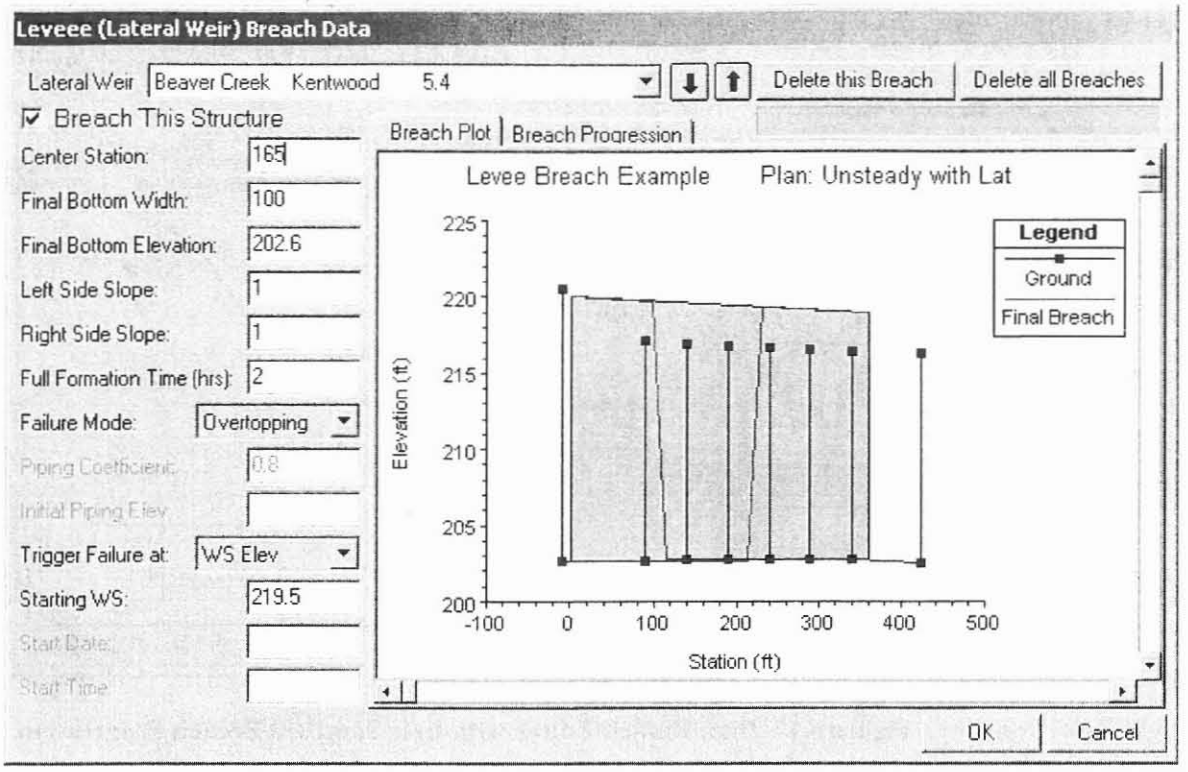


Figure 8.18 Levee Breaching Editor.

As shown in Figure 8.18, this editor contains the same information as the Dam Breach editor. For a description of the variables please review the section on Dam Breaching above. More detailed information about levee breaching can be found in Chapter 16 of this manual.

Mixed Flow Options. This option allows the user to change the parameters that control the computations of mixed flow regime within the unsteady flow simulation. This option was described previously in this chapter, under the section titled **Computational Settings**. Please review that section of this chapter for the details of how to use the mixed flow regime option and controlling the parameters.

Calculation Options and Tolerances. This option allows the user to set some computation options and to override the default settings for the calculation tolerances. These tolerances are used in the solution of the momentum equation. **Warning !!!** - Increasing the default calculation tolerances could result in computational errors in the water surface profile. The tolerances are as follows:

Theta implicit weighting factor: This factor is used in the finite difference solution of the unsteady flow equations. The factor ranges between 0.6 and 1.0. A value of 0.6 will give the most accurate solution of the equations, but is more susceptible to instabilities. A value of 1.0 provides the most stability in the solution, but may not be as accurate for some data sets. The default

value is set to 1.0. Once the user has the model up and running the way they want it, they should then experiment with changing theta towards a value of 0.6. If the model remains stable, then a value of 0.6 should be used. In many cases, you may not see an appreciable difference in the results when changing theta from 1.0 to 0.6. However, every simulation is different, so you must experiment with your model to find the most appropriate value.

Water surface calculation tolerance: This tolerance is used to compare the difference between the computed and assumed water surface elevations at cross sections. If the difference is greater than the tolerance, the program continues to iterate for the current time step. When the difference is less than the tolerance, the program assumes that it has a valid numerical solution. The default value is set to 0.02 feet.

Storage area elevation tolerance: This tolerance is used to compare the difference between computed and assumed water surface elevations at storage areas. If the difference is greater than the tolerance, the program continues to iterate for the current time step. When the difference is less than the tolerance, the program can go on to the next time step. The default tolerance for storage areas is set to 0.1 feet.

Maximum number of iterations: This variable defines the maximum number of iterations that the program will make when attempting to solve the unsteady flow equations using the specified tolerances. The default value is set to 20, and the allowable range is from 0 to 40.

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 previous paragraph, 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 interpolation: 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 interpolated time steps: This option defines the

maximum number of interpolated time steps that the program can use during time slicing, as described in the previous paragraph.

Weir flow stability factor: This factor is used to increase the stability of the numerical solution in and around a weir. This 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 oscillations in the computed weir flow. The default value is 1.0. If you observe oscillations in the computed flow over the weir, you should first check to see if you are using a small enough computation interval. If the computation interval is sufficiently small, you should then try increasing this coefficient to see if it solves the problem.

Spillway flow stability factor: This factor is used to increase the stability of the numerical solution in and around a gated spillway. This 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 oscillations in the computed spillway flow. The default value is 1.0. If you observe oscillations in the computed flow over the spillway, you should first check to see if you are using a small enough computation interval. If the computation interval is sufficiently small, you should then try increasing this coefficient to see if it solves the problem.

Weir flow submergence decay exponent: This coefficient is used to stabilize the solution of flow over a weir for highly submerged weirs. This factor varies from 1.0 to 3.0. As the headwater and tailwater stages become closer together, occasionally oscillations in the solution can occur. This exponent will prevent this from happening. The default value of one has no effect. As you increase the coefficient, dampening of the oscillations will occur. See the section called Model Accuracy, Stability, and Sensitivity later in this chapter for greater detail on this factor.

Spillway flow submergence decay exponent: This coefficient is used to stabilize the solution of flow over a gated spillway for highly submerged flows. This factor varies from 1.0 to 3.0. As the headwater and tailwater stages become closer together, occasionally oscillations in the solution can occur. This exponent will prevent this from happening. The default value of 1.0 has no effect. As you increase the coefficient, dampening of the oscillations will occur. See the section called Model Accuracy, Stability, and Sensitivity later in this chapter for greater detail on this factor.

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 bridge sections.

Output Options. This option allows the user to set some additional output flags. The following is a list of the available options:

Write Initial Conditions file: This option allows the user to write out a "Hot Start" file. A hot start file can be used to set the initial conditions of the system for a subsequent run. This is commonly done in real time forecasting, where you want to use the results at a specific time from a previous run to be the initial conditions of the next run. The user is require to enter a time in hours from the beginning of the current simulation, which represents the time at which the conditions of the system will be written to the "Hot Start" file.

Write Detailed Output for Debugging: This option allows the user to turn on detailed output that is written to a log file. The user has the option to set a specific time window in which the program will only output information within this time. This option is used when there is a problem with the unsteady flow solution, in that it may be oscillating or going completely unstable. When this occurs, the user should turn this option on and re-run the program. After the run has either finished or blown up, you can view the log file output by selecting **View Computation Log File** from the **Options** menu of the Unsteady Flow Simulation window. This log file will show what is happening on a time step by time step basis. It will also show which cross section locations the program is having trouble balancing the unsteady flow equations, as well as the magnitude of the errors.

Check Data Before Execution. This option provides for comprehensive input data checking. When this option is turned on, data checking will be performed when the user presses the compute button. If all of the data are complete, then the program allows the unsteady flow computations to proceed. If the data are not complete, or some other problem is detected, the program will not perform the unsteady flow analysis, and a list of all the problems in the data will be displayed on the screen. If this option is turned off, data checking is not performed before the unsteady flow execution. The default is that the data checking is turned on.

View Computation Log File. This option allows the user to view the contents of the unsteady flow computation log file. The interface uses the Windows Notepad program to accomplish this. The log file contains detailed information of what the unsteady flow computations are doing on a time step by time step basis. This file is very useful for debugging problems with your unsteady flow model.

Saving The Plan Information

To save the Plan information to the hard disk, select **Save Plan** from the **File** menu of the simulation window. Whenever any option is changed or modified on the Unsteady Flow Analysis window, the user should Save the Plan.

Starting the Computations

Once all of the data have been entered, and a Plan has been defined, the unsteady flow computations can be performed by pressing the **Compute** button at the bottom of the Unsteady Flow Simulation window. When the compute button is pressed, a separate window will appear showing you the progress of the computations (Figure 8.19). The information that appears in the window is there as an indicator of the programs progress during the computations, and to display any computational messages. When the computations have been completed, the user can close the computations window by clicking the upper right corner of the window, or the close button at the bottom. If the computations ended normally (i.e. all of the processes ran with no error messages), then the user can begin to look at the output. If the program does not finish normally, then the user should turn on the detailed log file output option and re-run the program. Then view the log file output to begin debugging the problem.

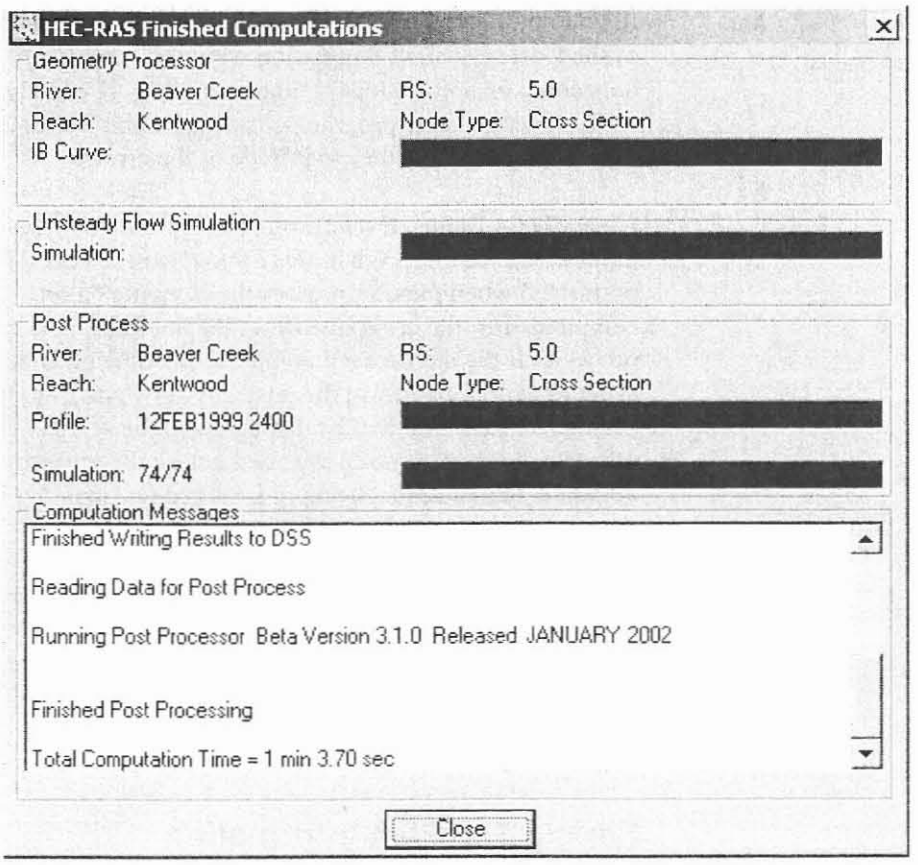


Figure 8.19 Unsteady Flow Computations Window

Calibration of Unsteady Flow Models

Calibration is the adjustment of a model's parameters so that it reproduces observed data to an acceptable accuracy. The following is a list of common problems and factors to consider when calibrating an unsteady flow model.

Observed Hydrologic Data

Stage Records. In general, measured stage data is our most accurate hydrologic data. Measured stage data is normally well within +/- 1.0 feet of accuracy. However, errors can be found in measured stage data. Some common problems are:

1. The gages float gets stuck at a certain elevation during the rise or fall of the flood wave.

2. The recorder may systematically accumulate error over time.

3. The gage reader of a daily gage misses several days and guesses at the stage recordings.

4. There is an error in the datum of the gage.

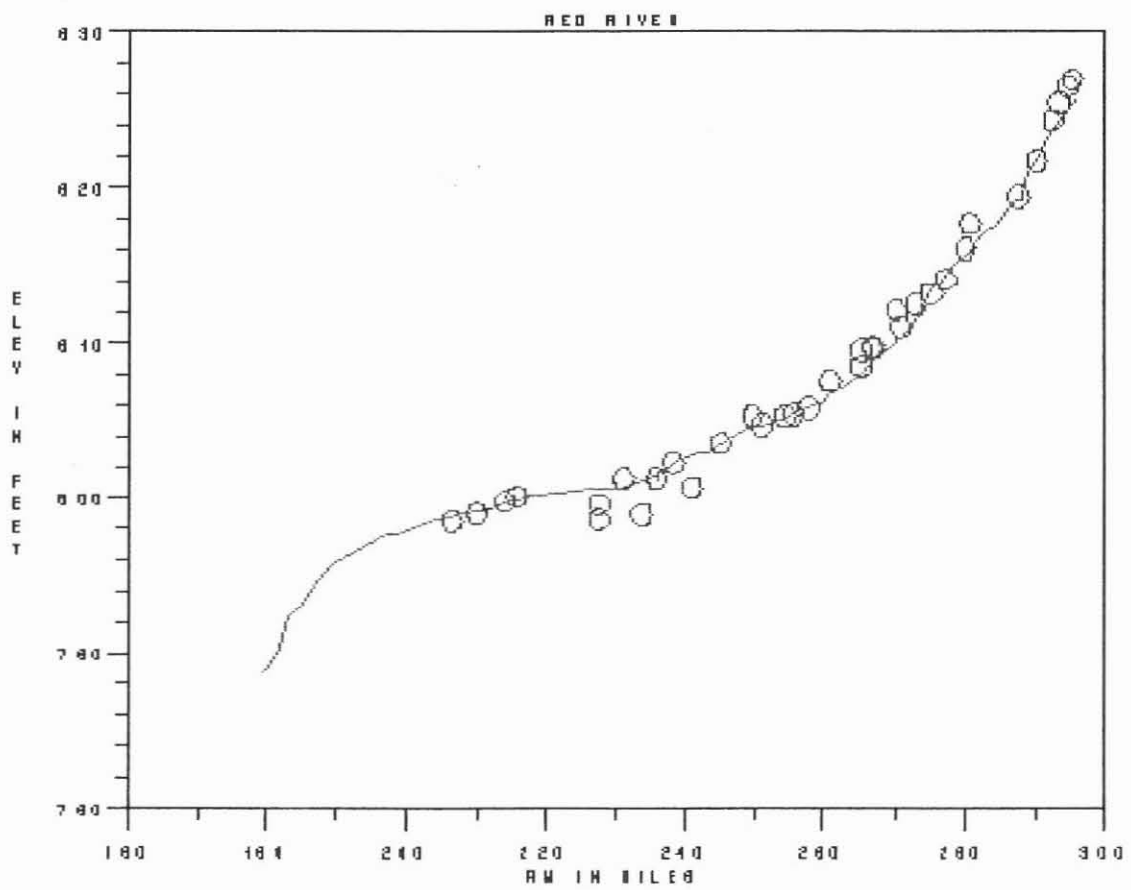
Flow Records. Flow records are generally computed from observed stages using single valued rating curves. These rating curves are a best fit of the measured data. The USGS classifies good flow measurements from Price current meters to be within ±5% of the true value. Some believe that this assumed error is optimistic. In any case, ±5%, on many river systems, translates into a stage error of ±1 foot. Acoustic velocity meters (AVM) provide a continuous record, 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 almost always suspect. In general it is very difficult to get accurate velocity measurements using a price current meter from a boat. 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 depict the quality of the published flow data is to compare the measured flow values to the currently published rating curve. However, if the flow 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 for large streams are in mean daily flow. The modeler must somehow assign time values to these records.

High Water Marks. High water marks are estimated from the upper limit of stains and debris deposits found on buildings, bridges, trees, and other structures. Wind and wave actions can cause the debris lines to be higher than the actual water surface. Capillary action can cause stains on buildings to migrate upward, depending on the material used for the building walls. High water marks in the overbank area are often higher than in the channel. The overbank water is moving slower and may be closer to the energy gradeline. High water marks on bridge piers are often equal to the energy gradeline, not the average water surface. This is due to the fact that the water will run up the front of the pier to an elevation close to the energy gradeline.

Shown in the Figure 8.20 below is a comparison between high water marks and the computed maximum water surface profile. Note the scatter in the high water marks, particularly around river station 230. Which mark is



accurate?

Figure 8.20 Computed Water Surface Profile Versus Observed High Water Marks.

Ungaged Drainage Area. For an unsteady flow model to be accurate, it must have flow input from all of the contributing area. In many studies a significant portion of the area is ungaged. Discharge from ungaged areas can be estimated from either hydrologic models or by taking flow from a gaged watershed with similar hydrologic characteristics and multiplying it by a simple drainage area ratio.

An example of accounting for ungaged drainage area is shown below for the Red River of the North.

Stream	Station	River Mile	Gaged Drainage (Sq. Miles)
Red River	Grand Forks	296	30,100
Turtle River	Manvel	272.9	613
Forest River	Minto	242.5	740
Snake River	Alvarado	229.9	309
Middle River	Argyle	9.72	265
Park River	Grafton	221.9	695
	Total of Gaged Tributaries		2,622
Red River	Drayton	206.7	34,800
		Total Ungaged	2,078

Stream	River Mile	Ungaged Drainage (Sq. Miles)	Pattern Hydrograph	Drainage Area Ratio
Grand Marais Creek	288.6	298	Middle River	1.12
Tamarac River	218.5	320	Middle River	1.21
Remaining		1,460	Middle River	5.51

Figure 8.21 Example Drainage Area Accounting for Red River of the North.

As shown in Figure 8.21, ungaged areas can be accounted for by using a pattern hydrograph of a similar watershed (Middle River), then calculating a drainage area ratio of contributing areas (Ungaged area divided by pattern hydrograph area).

River and Floodplain Geometry

It is essential to have an adequate number of cross sections that accurately depict the channel and overbank geometry. This can be a great source of

error when trying to calibrate. Additionally, all hydraulic structures must be accurately depicted. Errors in bridge and culvert geometry can be significant sources of error in computed water surface profiles. Another important factor is correctly depicting the geometry at stream junctions (flow combining and splitting locations). This is especially important at flow splits, and areas in which flow reversals will occur (i.e. flow from a main stem backing up a tributary).

Also, a one-dimensional model assumes a constant water surface in each cross section. For some river systems, the water surface may vary substantially between the channel and the floodplain. If this is the case in your model, it may be necessary to separate the channel and the floodplain into there own reaches or model the overbank area as a series of storage areas.

Roughness Coefficients

Roughness coefficients are one of the main variables used in calibrating a hydraulic model. Generally, for a free flowing river, roughness decreases with increased stage and flow (Figure 8.22). However, if the banks of a river are rougher than the channel bottom (do to trees and brush), then the composite n value will increase with increased stage. Sediment and debris can also play an important role in changing the roughness. More sediment and debris in a river will require the modeler to use higher n values in order to match observed water surfaces.

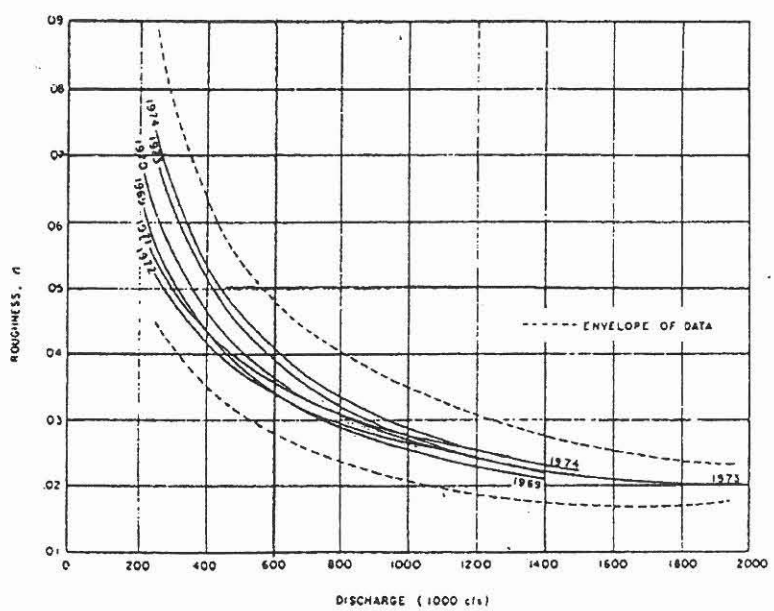


Figure 8.22 Roughness Versus Discharge for the Mississippi River at Arkansas City.

Looped Rating Curves. Excluding cataclysmic events such as meander cutoffs or a new channel, the river will pass any given flow within a range of stages. The shift in stage is a result of the following: shifts in channel bedforms; the dynamics of the hydrograph (how fast the flood wave rises and falls); backwater (backwater can significantly change the stage at a given cross section for a given flow); and finally, the slope of the river (flatter streams tend to have greater loops in the rating curve). Figure 8.23 below shows a looped rating for a single event. Generally, the lower stages are associated with the rising side of a flood wave, and the higher stages are associated with the falling side of the flood wave.

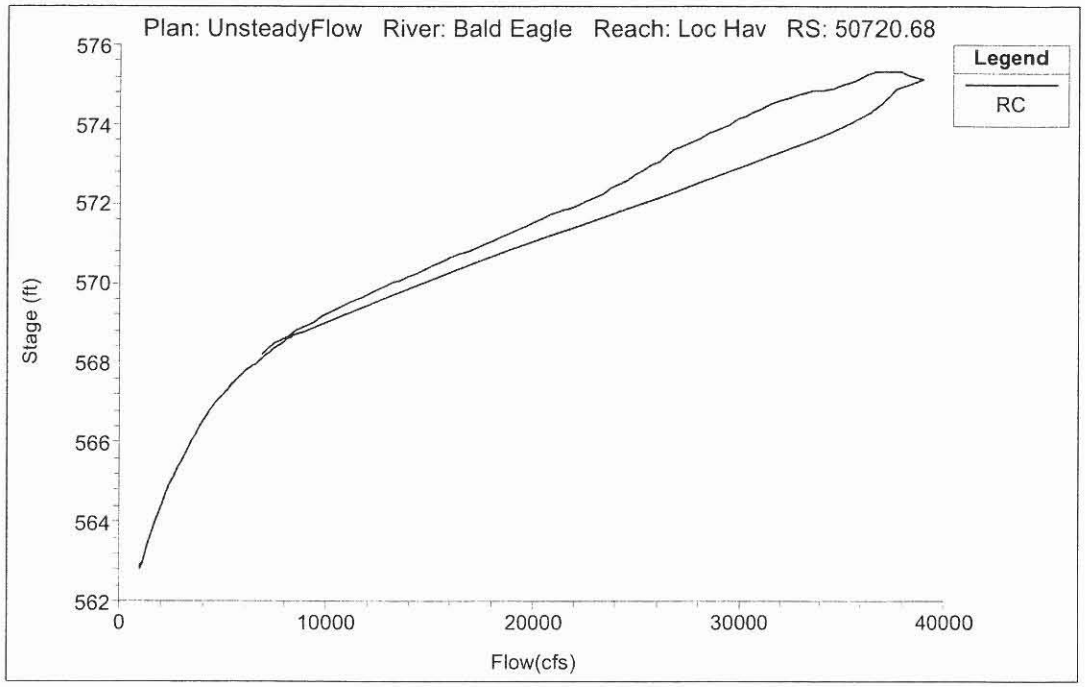


Figure 8.23 Looped Rating Curve Example

Alluvial Rivers. In an alluvial stream the channel boundary, as well as the meandering pattern of the stream, are continuously being re-worked by the flow of water. Alluvium is unconsolidated granular material, which is deposited by flowing water. An alluvial river is incised into these alluvial deposits. The flow characteristics of the stream are defined by the geometry and roughness of the cross-section below the water surface. The reworking of the cross section geometry and meander pattern is greatest during high flow, when the velocity, depth of water, and sediment transport capacity are the greatest. For some streams, which approach an equilibrium condition, the change in morphology (landforms) is small. For other streams, the change in morphology is much larger. The change can be manifest as changes in roughness or a more dynamic change such as the cut-off of a meander loop, which shortens the stream and starts a process which completely redefines the bed.

A typical meandering river is shown in Figure 8.24 below. Pools are at the outside of bends, and a typical pool cross-section is very deep. On the inside of the bend is a point bar. Crossings are between the meander bends. A typical crossing cross-section is much shallower and more rectangular than a pool cross-section.

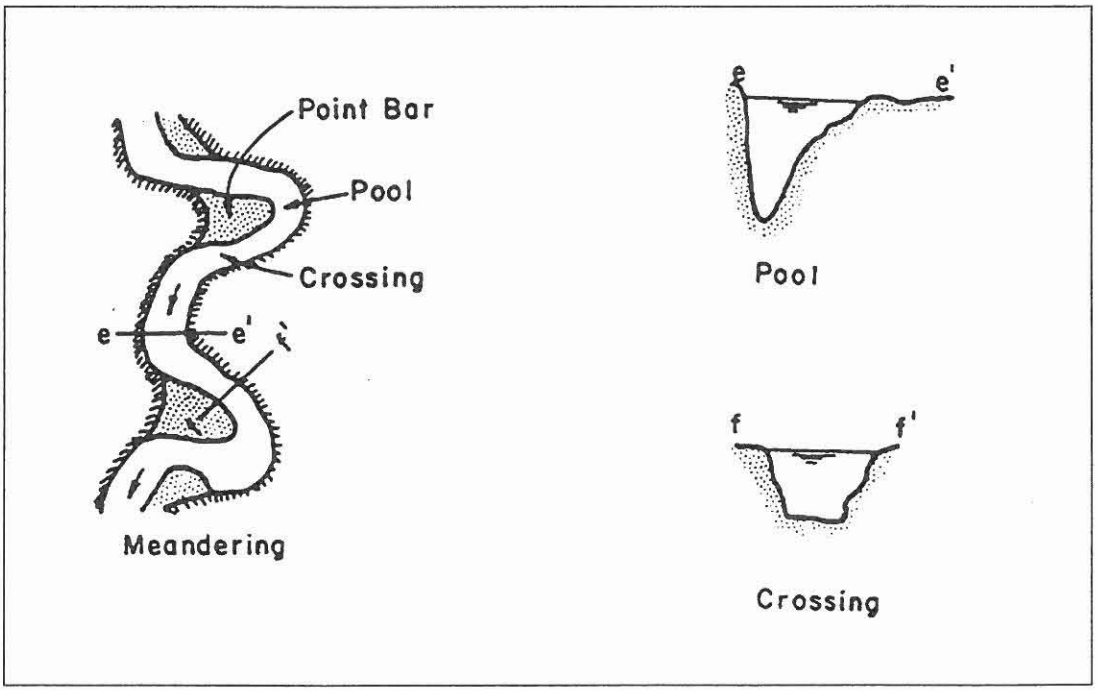


Figure 8.24 Morphology of a Meandering River

An invert profile for the Mississippi River is shown in the Figure 8.25. Note the pools and crossings. The water surface profile is controlled by the crossing cross-sections, particularly at low flow. The conveyance properties of pool cross-sections are only remotely related to the water surface. This poses a significant problem when calibrating a large river.

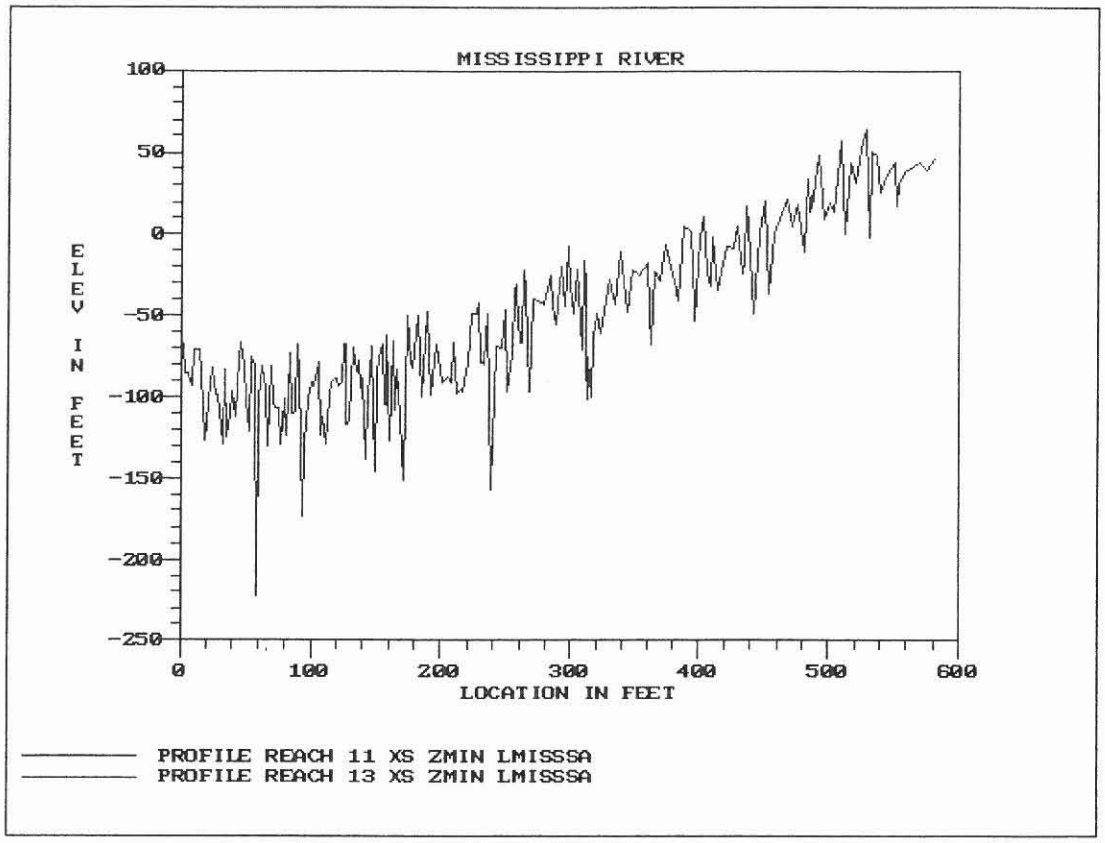


Figure 8.25 Invert Profile for Lower Mississippi River

As stage and flow increase you have an increase in stream power (stream power is a function of hydraulic radius, slope, and velocity). The bed forms in an alluvial stream tend to go through the following transitions:

1. Plane bed without sediment movement.
2. Ripples.
3. Dunes.
4. Plane bed with sediment movement.
5. Anti-dunes.
6. Chutes and pools.

Generally, anti-dunes and chutes and pools are associated with high velocity streams approaching supercritical flow. The bed form process is shown graphically in Figure 8.26.

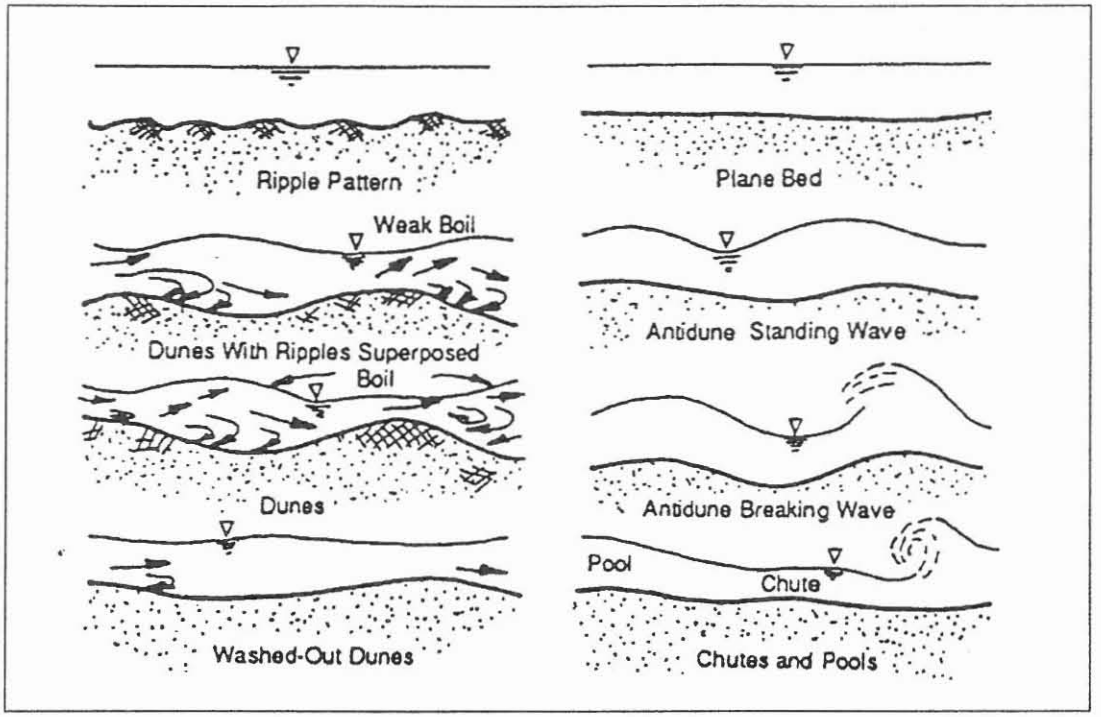


Figure 8.26 Transitions of Bed Forms in an Alluvial Stream

Typical Manning's roughness coefficients for the different bed forms presented above are shown in the following table:

Table 8-1 Roughness Variations for Alluvial Streams

Bed Forms	Range of Manning's n
Ripples	0.018 – 0.030
Dunes	0.020 – 0.035
Washed Out Dunes	0.014 – 0.025
Plane Bed	0.012 – 0.022
Standing Waves	0.014 – 0.025
Antidunes	0.015 – 0.031

Note: This table is from the book "Engineering Analysis of Fluvial Streams", by Simons, Li, and Associates.

Bed forms also change with water temperature. Because water is more viscous at lower temperatures, it becomes more erosive, reducing the height and the length of the dunes. At higher temperatures, when the water is less viscous, the dunes are higher and of greater length. Since the larger dunes are more resistant to flow, the same flow will pass at a higher stage in the summer than in the winter. Larger rivers such as the Mississippi River and the Missouri River show these trends. Figure 8.27 shows the seasonal shift for the Mississippi River at St. Louis.

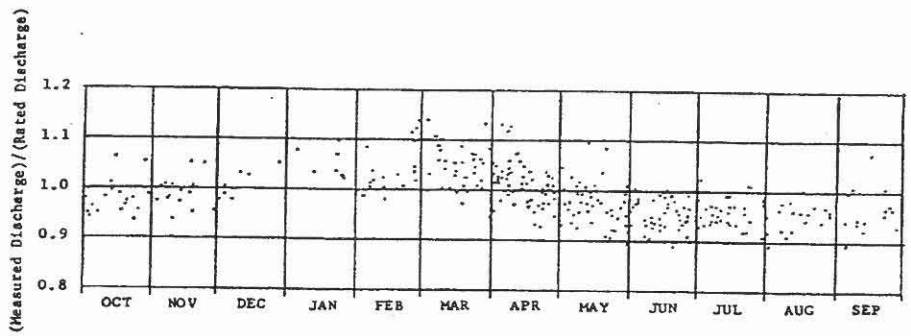


Figure 8.27 Changes in Roughness due to Temperature in The Mississippi River.

River and Floodplain Storage

Cross Sectional Storage. The active flow area of a cross section is the region in which there is appreciable velocity. This part of the cross section is conveying flow in the downstream direction. Storage is the portion of the cross section in which there is water, but it has little or no velocity. Storage can be modeled within a cross section by using the ineffective flow area option in HEC-RAS. The water surface elevation within the cross section storage is assumed to have the same elevation as the active flow portion of the cross section.

The storage within the floodplain is responsible for attenuating the flood hydrograph and, to some extent, delaying the flood wave.

Effects of Overbank Storage. Water is taken out of the rising side of the flood wave and returned on the falling side. An example of the effects of overbank storage is shown in Figure 8.28. In this example, the water goes out into storage during the rising side of the flood wave, as well as during the peak flow. After the peak flow passes, the water begins to come out of the storage in the overbank and increases the flow on the falling side of the floodwave.

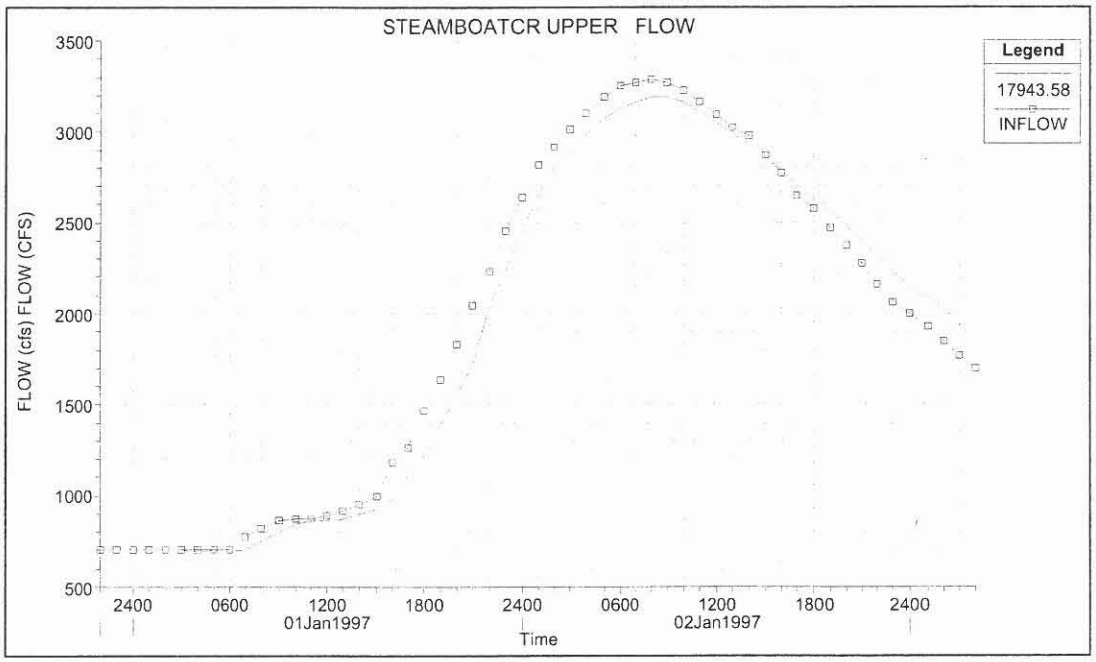


Figure 8.28 Example of The Effects of Overbank Storage

Off Line Storage. Off line storage is an area away from the main river in which water can go from the main river to the ponding area. The connection between the ponding area and the river may be a designed overflow, or it may just be a natural overflow area. The water in the ponding area is often at a different elevation that the main river, therefore, it must be modeled separately from the cross sections describing the main river and floodplain. Within HEC-RAS, ponding areas are modeled using what we call a storage area. Storage areas can be connected hydraulically to the river system by using a lateral weir/spillway option in HEC-RAS.

The effect that off line storage has on the hydrograph depends on the available volume and the elevation at which flow can get into and out of the storage area. Shown in Figure 8.29 is an example of an off-line storage area that is connected to the river through a lateral weir/spillway. The flow upstream and downstream of the offline storage area remains the same until the water surface elevation gets higher than the lateral weir. Water goes out into the lateral storage facility the whole time it is above the weir (i.e. the storage area elevation is always lower than the river elevation in this example). This continues until later in the event, when the river elevation is

below the lateral weir and flow can no longer leave the river. In this example, the flow in the storage area does not get back into the river system until much later in the event, and it is at a very slow rate (possible drained by culverts to a downstream location).

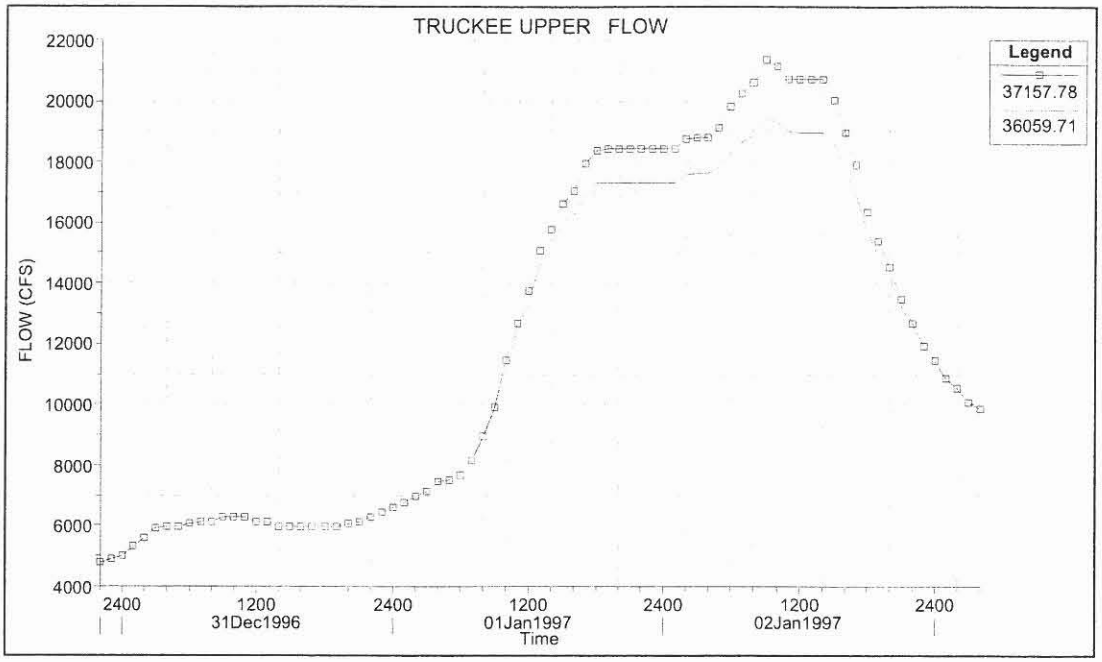


Figure 8.29 Example effects of off-line Storage

Hydraulic Structure Coefficients

Bridges and culverts tend to have a local effect on stage, and a minimum affect on the flow hydrograph (this depends on the amount of backwater they cause). The coefficients that are important in bridge modeling are: Manning's n values; contraction and expansion coefficients; pier loss coefficients, and pressure and weir flow coefficients for high flows. Culvert hydraulics are dependent upon the size of the culverts and shape of the entrance. Additional variables include Manning's n values and contraction and expansion coefficients.

The effects of Inline weirs/spillways can be substantial on both the stage and the flow attenuation of the hydrograph. The effects on the hydrograph will depend upon the available storage volume in the pool upstream of the structure, as well as how the structure is operated. Lateral weir/spillway structures can have a significant impact on the amount of water leaving the river system. Therefore gate and weir coefficients for these structures can be extremely critical to getting the right amount of flow leaving the system.

Steps To Follow in The Calibration Process

The following is a general list of steps to follow when calibrating an unsteady flow model:

1. Run a range of discharges in the Steady-Flow mode, and calibrate n values to established rating curves at gages and known high water marks.

2. Select specific events to run in unsteady flow mode. Ensure each event goes from low flow to high flow, and back to low flow.

3. Adjust storage and lateral weirs to get good reproduction of flow hydrographs (Concentrate on timing, peak flow, volume, and shape).

4. Adjust Manning's n values to reproduce stage hydrographs.

5. Fine tune calibration for low to high stages by using "Discharge-Roughness Factors" where and when appropriate.

6. Further refine calibration for long-term modeling (period of record analysis) with "Seasonal Roughness Factors" where and when appropriate.

7. Verify the model calibration by running other flow events or long term periods that were not used in the calibration.

8. If further adjustment is deemed necessary from verification runs, make adjustments and re-run all events (calibration and verification events).

General Trends When Adjusting Model Parameters

In order to understand which direction to adjust model parameters to get the desired results, the following is a discussion of general trends that occur when specific variables are adjusted. These trends assume that all other geometric data and variables will be held constant, except the specific variable being discussed.

Impacts of Increasing Manning's n. When Manning's n is increased the following impacts will occur:

1. Stage will increase locally in the area where the Manning's n values were increased.

2. Peak discharge will decrease (attenuate) as the flood wave moves downstream.

3. The travel time will increase.

4. The loop effect will be wider (i.e. the difference in stage for the same flow on the rising side of the flood wave as the falling side will be greater). An example of this is shown in Figure 8.30.

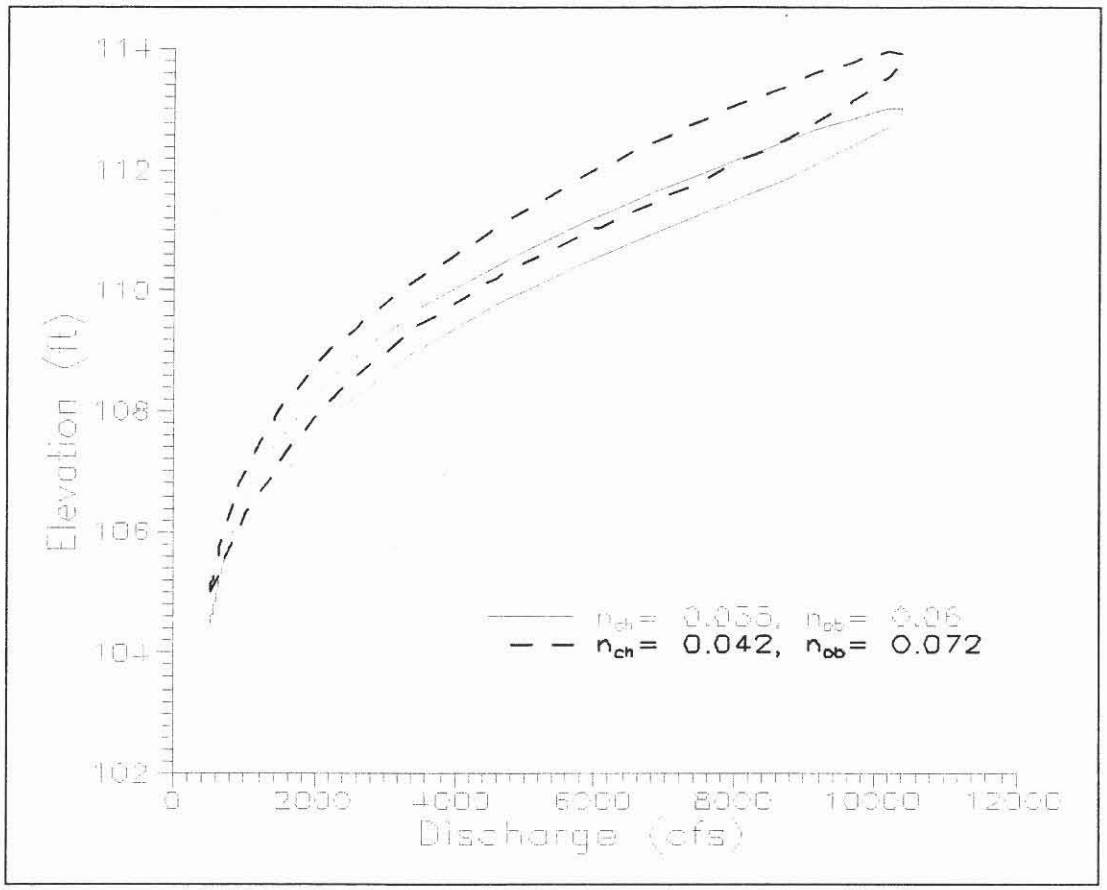


Figure 8.30 Example of Wider Loop for Higher Manning's n

Impacts of Increasing Storage. When storage within the floodplain is increased, the following impacts will occur:

1. Peak discharge will decrease as the flood wave moves downstream.

2. The travel time will increase.

3. The tail of the hydrograph will be extended.

4. The local stage (in the area of the increased storage) may increase or decrease. This depends upon if you are trading conveyance area for storage area, or just simply increasing the amount of storage area.

Calibration Suggestions and Warnings

The following is a list of suggestions and warnings to consider when calibrating an unsteady flow model:

1. Calibrate mostly to stages. Flow data is derived from stage. Be wary of discharge derived from stage using single value rating curves.

2. Do not force a calibration to fit with unrealistic Manning's n values or storage.

3. If using a single-valued rating curve at the downstream boundary, move it far enough downstream so it doesn't effect the results in the study reach.

4. Discrepancies may arise from a lack of quality cross-section data.

5. The volume of off-channel storage areas is often underestimated, which results in a flood wave that travels to fast.

6. Be careful with old HEC-2 and RAS studies done for steady flow only. The cross sections may not depict the storage areas. Defining storage is not a requirement for a steady flow model.

7. Calibration should be based on floods that encompass a wide range of flows, low to high.

8. For tidally influenced rivers and flows into reservoirs, the inertial terms in the momentum equation are very important. Adjusting Manning's n values may not help. Check cross sectional shape and storage.

9. You must be aware of any unique events that occurred during the flood. Such as levee breaches and overtopping.

Model Accuracy, Stability, and Sensitivity

This section of the manual discusses model accuracy, stability, and sensitivity. In order to develop a good unsteady flow model of a river system, the user must understand how and why the solution of the unsteady flow equations becomes unstable. This knowledge will help you figure out why your particular model may be having stability problems. Additionally, it is important to understand the trade-offs between numerical accuracy (accurately solving the equations) and model stability. Finally, model sensitivity will be discussed in order to give you an understanding of what parameters affect the model and in what ways.

Model Accuracy

Model accuracy can be defined as the degree of closeness of the numerical solution to the true solution. Accuracy depends upon the following:

1. Assumptions and limitations of the model (i.e. one dimensional model, subcritical flow only for unsteady flow).

2. Accuracy of the geometric Data (cross sections, Manning's n values, bridges, culverts, etc…).

3. Accuracy of the flow data and boundary conditions (inflow hydrographs, rating curves, etc…).

4. Numerical Accuracy of the solution scheme (solution of the unsteady flow equations).

Numerical Accuracy. If we assume that the 1-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. Finite difference solutions are approximate. An exact solution of the equations is not feasible for complex river systems, so HEC-RAS uses an implicit finite difference scheme.

Model Stability

An unstable numerical model is one for which certain types of numerical errors grow to the extent at which the solution begins to oscillate, or the errors become so large that the computations can not continue. This is a common problem when working with an unsteady flow model of any size or complexity. The following factors will affect the stability and numerical accuracy of the model:

1. Cross section spacing.

2. Computation time step.

3. Theta weighting factor for numerical solution.

4. Solution iterations.

5. Solution tolerances.

6. Weir and spillway stability factors.

7. Weir and spillway submergence factors.

Cross-Section Spacing. Cross sections should be placed at representative locations to describe the changes in geometry. 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.

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. However, most streams lie some where in between these two spacing distances.

The general question about cross section spacing is "How do you know if you have enough cross sections." The easiest way to tell is to add additional cross sections (this can be done through the HEC-RAS cross section interpolation option) and save the geometry as a new file. Then make a new plan and execute the model, compare the two plans (with and without interpolated cross sections). If there are no significant differences between the results (profiles and hydrographs), then the original model without the additional cross sections is ok. If there are some significant differences, then additional cross sections should be gathered in the area where the differences occur. If it is not possible to get surveyed cross sections, or even cross sections from a GIS, then use the HEC-RAS interpolated cross sections. However, at least check the reasonableness of the interpolated cross sections with a topographic map. Edit any cross sections that do not look reasonable.

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

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

Therefore:

$$\Delta t \leq \frac{\Delta x}{V_w}$$

Where: V_w	=	Flood wave speed, which is normally greater than the average velocity.
V	=	Average velocity of the flow.
Δx	=	Distance between cross sections.
Δt	=	Computational time step.

For most rivers the flood wave speed can be calculated as:

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

However, an approximate way of calculating flood wave speed is to multiply the average velocity by a factor. Factors for various channel shapes are shown in the table below.

Table 8-2 Factors for Computing Wave Speed From Average Velocity

Channel Shape	Ratio Vw/V
Wide Rectangular	1.67
Wide Parabolic	1.44
Triangular	1.33
Natural Channel	1.5

Practical Time Step Selection. For medium to large rivers, the Courant condition may yield time steps that are too restrictive (i.e. a larger time step could be used and still maintain accuracy and stability). A practical time step is:

$$\Delta t \leq \frac{Tr}{20}$$

Where: T_r = Time of rise of the hydrograph to be routed.

However, you may need to use a smaller time step when you have lateral weirs/spillways and hydraulic connections between storage areas and the river system. Also, if you are opening and closing gates quickly, or modeling a Dam or Levee breach, you will need to use very small time steps (less than a minute, maybe even as low as 5 seconds).

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

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, errors in the solution 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. Test the sensitivity of theta to your data set. If reducing theta does not change the solution, then the larger value should be used to insure greater stability.

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

Solution Tolerances. Two solution tolerances can be set or changed by the user: water surface calculation (0.02 default) and Storage area elevation (0.10 default). The default values should be good for most river systems. Only change them if you are sure! Making the tolerances larger can reduce the stability of the solution. Making them smaller can cause the program to go to the maximum number of iterations every time.

Weir and Spillway Stability Factors. Weirs and Spillways can often be a source of instability in the solution. Especially lateral structures, which take flow away or bring it into the main river. During each time step, the flow over a weir/spillway is assumed to be constant. This can cause oscillations by sending too much flow during a time step. One solution is to reduce the time step. Another solution is to use Weir and spillway stability factors, which can smooth these oscillations by damping the computed flows. However, using these stability factors can reduce the accuracy of the computed values.

The weir and spillway stability factors can range from 1.0 to 3.0. The default value of 1.0 is essentially no damping of the computed flows. As you increase the factor you get greater dampening of the flows (which will provide for greater stability), but less accuracy.

Weir and Spillway Submergence Factors. When you have a weir or spillway connecting two storage areas, or a storage area and a reach, oscillations can occur when the weir or spillway becomes highly submerged. The program must always have flow going one way or the other when the water surface is above the weir/spillway. When a weir/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 weir, 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.

To reduce the oscillations, the user can increase the Weir/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 1.0 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 Figure 8.31.

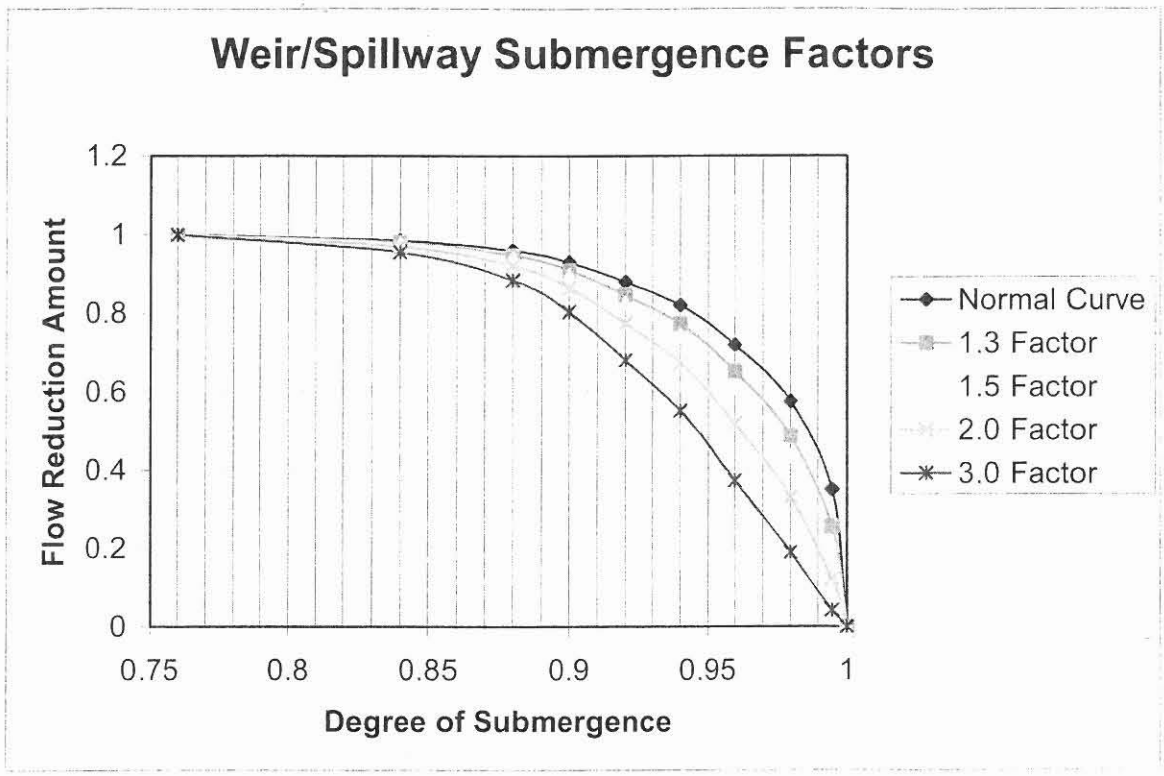


Figure 8.31 Weir/Spillway Submergence Factors

Common Stability Problems. The following is a list of some of the most common reasons for instabilities occurring in an unsteady flow model like HEC-RAS:

Too large of a time step: When the solution scheme solves the unsteady flow equations, derivatives are calculated with respect to distance and time. If the changes in hydraulic properties at a give cross section are changing rapidly with respect to time, the program will go unstable. The solution to this problem is to decrease the computational 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. In situations where the flow area, depth, and/or velocity are changing rapidly, it is necessary to add additional cross sections to keep the solution stable.

Model goes to critical depth: The default solution methodology for unsteady flow routing within HEC-RAS is generally for subcritcal flow. However, the software does have an option to run in a mixed flow regime mode. This option should not be used unless you truly believe you have a mixed flow regime river system. If you are running the software in the default mode (subcritical only, no mixed flow), and if the program goes down to critical depth at a cross section, the changes in area, depth, and velocity are very high. This sharp increase in the water surface slope will often cause the

program to overestimate the depth at the next cross section upstream, and possible underestimate the depth at the next cross section downstream (or even the one that went to critical depth the previous time step). One solution to this problem is to increase the Manning's n value in the area where the program is first going to critical depth. This will force the solution to a subcritical answer and allow it to continue with the run. Obviously, this is creating a solution that may be higher than the true solution, but it may be necessary in order to allow the program to continue with the run. If you feel that the true water surface should go to critical depth, or even to a supercritical flow regime, then the mixed flow regime option should be turned on.

Bad downstream boundary condition: If the user entered downstream boundary condition causes abrupt jumps in the water surface, or water surface elevations that are too low (approaching or going below critical depth), this can cause oscillations in the solution that may lead to it going unstable and stopping. Examples of this are rating curves with not enough points or just simply to low of stages; and normal depth boundaries where the user has entered to 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 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 a 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. Permanent ineffective flow areas allow water to convey over top of the ineffective area, so the change in conveyance and area is small. The use of high Manning's n values reduces the abruptness in the change in area and conveyance when the ineffective flow area gets turned off and starts conveying water.

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. With this number of points, this problem should not happen.

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. One free flow curve (headwater versus flow, with no influence from the tailwater) is calculated with fifty points to define it, then up to fifty submerged curves (headwater versus flow, staring at a particular tailwater) are calculated with up to 20 points to define each curve. The user can control how many submerged curves get calculated, how many points in each curve, 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 entering specific table limits for maximum headwater, tailwater, flow, and head difference.

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

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, possibly even producing a negative depth. This is due 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.

Detecting Stability Problems. One of the hardest things about using an unsteady flow model is to get the model to be stable, as well as accurate, for the range of events to be modeled. When you first start putting together an unsteady flow model, undoubtedly you will run in to some stability problems. The question is, how do you know you are having a stability problem? The following is a list of stability problem indicators:

1. Program stops running during the simulation with a math error, or states that the matrix solution went unstable.

2. Program goes to the maximum number of iterations for several time steps in a row (this is not always a stability problem).

3. There are oscillations in the computed stage and flow hydrographs, or the water surface profiles.

If you detect a possible stability problem, in order to find the location of the problem you will need to turn on the detailed log output for debugging. Detailed log output is turned on by selecting **Options** and then **Output Options** from the Unsteady Flow Simulation manager. When this option is selected the following window will appear:

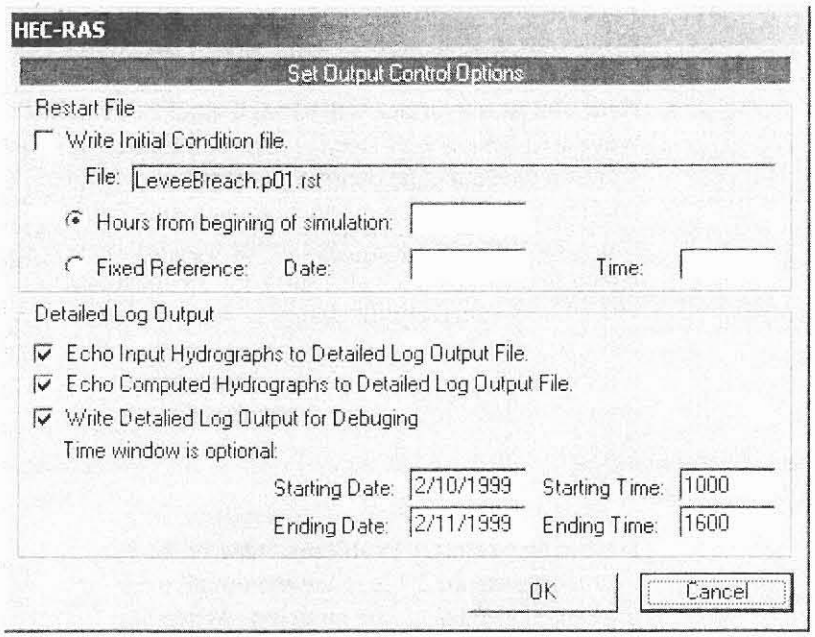


Figure 8.32 Detailed Log Output Control

As shown in Figure 8.32, the section at the bottom half of this editor is used for controlling the detailed log output. Three check boxes are listed. The first box can be used to turn on an echo of the hydrograph input to the model. This can be used to ensure that the model is receiving the correct flow data. The second check box can be used to turn on an echo of the computed hydrographs that will be written to the HEC-DSS. This is a good option for checking what was computed. However, if the user has selected to have hydrographs computed at many locations, this could end up taking a lot of file and disk space. The third check box is used to control the detailed output of results from the unsteady flow simulation. Selecting this options will cause the software to write detailed information on a time step by time step basis. This option is useful when the unsteady flow simulation is going unstable or completely blowing up (stopping). Checking this box turns on the detailed output for every time step. The user has the option to limit this output to a specific time window during the unsteady flow simulation. Limiting the log output is accomplished by entering a starting date and time and an ending date and time.

Viewing Detailed Log Output. 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).

The detailed log output file will contain the following output:

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

UNET Output: Detailed unsteady flow calculations including:

- Job control parameters
- Initial conditions calculations
- Detailed output for each time step

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

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. An example of the initial conditions output is shown in Figure 8.33 below.

LeveeBreach.bco - Notepad

File Edit Format Help

Initial Conditions from Backwater

RIVER NAME	REACH NAME	RIVER STATION	FLOW	WSEL	CRIT DEPTH	EG SLOPE	AREA
Beaver Creek	Kentwood	5.0	3000.00	209.30	207.02	0.0012071	2178.84
Beaver Creek	Kentwood	5.016	3000.00	209.40	207.13	0.0011585	2036.65
Beaver Creek	Kentwood	5.032	3000.00	209.49	207.23	0.0011494	1950.81
Beaver Creek	Kentwood	5.048	3000.00	209.58	207.30	0.0011500	1913.43
Beaver Creek	Kentwood	5.065	3000.00	209.67	207.36	0.0011672	1874.69
Beaver Creek	Kentwood	5.081	3000.00	209.75	207.40	0.0012010	1834.46
Beaver Creek	Kentwood	5.097	3000.00	209.82	207.41	0.0013485	1788.69
Beaver Creek	Kentwood	5.113	3000.00	209.92	207.46	0.0013873	1773.18
Beaver Creek	Kentwood	5.13	3000.00	210.03	207.46	0.0014094	1757.82
Beaver Creek	Kentwood	5.146	3000.00	210.17	207.90	0.0014358	1769.25
Beaver Creek	Kentwood	5.162	3000.00	210.31	208.27	0.0014840	1781.84
Beaver Creek	Kentwood	5.178	3000.00	210.45	208.61	0.0015081	1801.07
Beaver Creek	Kentwood	5.194	3000.00	210.60	208.86	0.0015066	1824.42
Beaver Creek	Kentwood	5.21	3000.00	210.75	209.17	0.0014833	1847.64
Beaver Creek	Kentwood	5.226	3000.00	210.89	209.50	0.0014471	1869.46
Beaver Creek	Kentwood	5.242	3000.00	211.02	209.68	0.0013911	1887.72
Beaver Creek	Kentwood	5.258	3000.00	211.15	209.74	0.0013163	1900.66
Beaver Creek	Kentwood	5.274	3000.00	211.26	209.76	0.0012231	1905.77
Beaver Creek	Kentwood	5.29	3000.00	211.37	209.67	0.0010797	1900.07
Beaver Creek	Kentwood	5.31	3000.00	211.36	210.30	0.0021558	1609.68
Beaver Creek	Kentwood	5.33	3000.00	211.38	210.22	0.0025166	1330.87
Beaver Creek	Kentwood	5.35	3000.00	211.48	209.15	0.0018303	1197.09
Beaver Creek	Kentwood	5.37	3000.00	211.60	208.10	0.0011982	1217.80
Beaver Creek	Kentwood	5.39	3000.00	211.70	207.21	0.0007646	1353.02
Beaver Creek	Kentwood	5.41	3000.00	211.77	207.22	0.0007778	1049.97
Beaver Creek	Kentwood	5.425	3000.00	211.77	208.41	0.0012525	898.97
Beaver Creek	Kentwood	5.44	3000.00	211.82	209.15	0.0016949	999.32
Beaver Creek	Kentwood	5.457	3000.00	211.96	209.44	0.0018008	1016.67
Beaver Creek	Kentwood	5.474	3000.00	212.11	209.77	0.0018442	1069.78
Beaver Creek	Kentwood	5.491	3000.00	212.29	210.14	0.0017970	1161.52
Beaver Creek	Kentwood	5.508	3000.00	212.46	210.99	0.0016890	1267.34
Beaver Creek	Kentwood	5.525	3000.00	212.61	211.22	0.0016485	1365.06
Beaver Creek	Kentwood	5.542	3000.00	212.76	211.29	0.0015495	1478.80
Beaver Creek	Kentwood	5.559	3000.00	212.90	211.25	0.0014375	1598.43

Figure 8.33 Example of Initial Conditions Output

During the unsteady flow computations, the program will output detailed information for cross sections, bridges/culverts, inline weir/spillways, lateral weir/spillways, storage areas, and storage area connections. This information should be reviewed closely when the software is having stability problems. An example of the detailed output for cross sections is shown in Figure 8.34 below.

beaver.bco - Notepad

File Edit Format Help

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

            Beaver Creek     Kentwood

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


                        Solving for T =     0.133

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

Figure 8.34 Detailed Output at a Cross Section.

When the program has stability problems, it will generally try to solve them by iterating. An example of a stability problem is shown in Figure 8.35. In this example the program did not solve the equations to the specified tolerances, and therefore it was iterating to improve the solution. As shown in Figure 8.35, the program iterated to the maximum number of iterations. At the end of the iterations a warning message states **"WARNING USED COMPUTED CHANGES IN FLOW AND STAGE AT MINIMUM ERROR. MINIMUM ERROR OCCURRED AT ITERATION XX."** This message means that the program could not solve the unsteady flow equations to the required tolerance within the specified number of iterations (default number of iterations is 20). Therefore it used the iteration that had the least amount of error in the numerical solution.

LeveeBreach.bco - Notepad

File Edit Format Help

```
                    Solving for T =     3.250

Iter  River          Station    Elev       DZ      Storage          Zsa        DZsa    River          Statio
  0 Beaver Creek     5.0        211.53   1.03961   STo #1           205.00   0.00000 Beaver Creek     5.0
  1 Beaver Creek     5.0        210.86  -0.95941   STo #1           205.00   0.00000 Beaver Creek     5.0
  2 Beaver Creek     5.0        210.22  -0.90790   STo #1           205.00   0.00000 Beaver Creek     5.0
  3 Beaver Creek     5.0        209.60  -0.89442   STo #1           205.00   0.00000 Beaver Creek     5.0
  4 Beaver Creek     5.0        208.96  -0.91305   STo #1           205.00   0.00000 Beaver Creek     5.0
  5 Beaver Creek     5.0        208.28  -0.96582   STo #1           205.00   0.00000 Beaver Creek     5.0
  6 Beaver Creek     5.0        207.56  -1.03982   STo #1           205.00   0.00000 Beaver Creek     5.0
  7 Beaver Creek     5.016      211.95   2.44811   STo #1           205.00   0.00000 Beaver Creek     5.0
  8 Beaver Creek     5.032      216.98   7.38552   STo #1           205.00   0.00000 Beaver Creek     5.0
  9 Beaver Creek     5.0        203.31  -1.34783   STo #1           205.00   0.00000 Beaver Creek     5.0
 10 Beaver Creek     5.162      218.46   7.18687   STo #1           205.00   0.00000 Beaver Creek     5.0
 11 Beaver Creek     5.0        198.49  -0.83206   STo #1           205.00   0.00000 Beaver Creek     5.0
 12 Beaver Creek     5.0        198.30  -0.26222   STo #1           205.00   0.00000 Beaver Creek     5.0
 13 Beaver Creek     5.0        198.25  -0.07942   STo #1           205.00   0.00000 Beaver Creek     5.0
 14 Beaver Creek     5.016      223.05  19.42409   STo #1           205.00   0.00000 Beaver Creek     5.0
 15 Beaver Creek     5.0        196.38  -0.88486   STo #1           205.00   0.00000 Beaver Creek     5.0
 16 Beaver Creek     5.0        195.82  -0.79409   STo #1           205.00   0.00000 Beaver Creek     5.0
 17 Beaver Creek     5.0        195.36  -0.66080   STo #1           205.00   0.00000 Beaver Creek     5.0
 18 Beaver Creek     5.065      210.10   3.19042   STo #1           205.00   0.00000 Beaver Creek     5.0
 19 Beaver Creek     5.081      210.03   1.40783   STo #1           205.00   0.00000 Beaver Creek     5.0
 20 Beaver Creek     5.0        193.35  -0.92478   STo #1           205.00   0.00000 Beaver Creek     5.0

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

                    Lateral Weirs

RIVER              STATION      QTOTAL      QWEIR     QGATE    SUBMERGENCE      Q PER CROSS SECTION
Beaver Creek       5.4            0.00       0.00      0.00     0.000          0.00        0.00        0.00

Hydrographs written at      3.25hours.

 Optimal time step for the next   15 minutes is      15.00 minutes; NSTEPS =

 THETA=1.000

      COMPUTED STAGES AND DISCHARGES AT T =      3.2500 HOURS -  2/10/1999 AT 0315 HOURS

                Beaver Creek      Kentwood
```

Figure 8.35 Example Detailed Time step Output for Cross Sections.

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

The variables that are printed out during the iterations are the following:

Iter = Iteration Number
River = The name of the river in which the largest stage error is occurring.
Station = River station with the largest error in the calculated stage.
ELEV = Computed water surface elevation at that river station
DZ = The "Numerical Error" in the computed stage at that location
Storage = Name of the storage area.
Zsa = Computed elevation of the storage area
Dzsa = The "Numerical Error" in the computed storage area elevation.
River = The name of the river in which the largest flow error is occurring.
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

After the iterations output, the program will show the computed stages and flows for all of the cross sections in which the user has selected to have

hydrographs computed. This is also useful information for detecting model stability problems. It is not always obvious as to which cross section or modeling component is causing the problem. Sometimes the program may blow up at one cross section, but the real problem is caused by a cross section upstream or downstream from this location. Detecting, finding, and fixing stability problems will require lots of experience to become proficient at it. Good luck, and don't get to discouraged!!!

Model Sensitivity

Model sensitivity is an important part of understanding the accuracy and uncertainty of the model. There are two types of sensitivity analysis that should be performed, Numerical Sensitivity and Physical Parameter Sensitivity.

Numerical Sensitivity. Numerical Sensitivity is the process of adjusting parameters that affect the numerical solution in order to obtain the best solution to the equations, while still maintaining model stability. The following parameters are typically adjusted for this type of sensitivity analysis:

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.

Weir/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.

Weir/Spillway Submergence Exponents – In general these parameters will not affect the answers significantly, they only provide greater stability when a spillway/weir is at a very high submergence. Try reducing them towards 1.0 (which is no factor) to see if the model will remain stable.

Physical Parameter Sensitivity. Physical Parameter Sensitivity is the process of adjusting hydraulic parameters and geometric properties in order to test the uncertainty of the models solutions. This type of sensitivity analysis is often done to gain an understanding of the possible range of solutions, given realistic changes in the model parameters. Another application of this type of sensitivity analysis is to quantify the uncertainty in the model results

for a range of statistical events (2, 5,10, 25, 50, 100 yr, etc…). The following data are often adjusted during this type of sensitivity analysis:

Manning's n Values – Manning's n values are estimated from physical data about the stream and floodplain. Sometimes Manning's n values are calibrated for a limited number of events. Either way, the values are not exact! The modeler should estimate a realistic range that the n values could be for their stream. For example, if you estimated an n value for a stream as 0.035, a realistic range for this might be 0.03 to 0.045. The modeler should run the lower Manning's n values and the higher Manning's n values to evaluate their sensitivity to the final model results.

Cross Section Spacing – Cross section spacing should always be tested to ensure that you have enough cross sections to accurately describe the water surface profiles. One way to test if you have enough cross sections is to use the HEC-RAS cross section interpolation routine, and interpolate enough cross sections to cut the average distance between cross sections in half. Re-run the model, if the results have not changed significantly, then your original model was probably fine. If the results do change significantly, then you should either get more surveyed cross sections or use the interpolated cross sections. If you use the interpolated cross sections, then you should at least look at a topographic map to ensure that the interpolated cross sections are reasonable. If the interpolated cross sections are not reasonable in a specific area, then simply edit them directly to reflect what is reasonable based on the topographic map.

Cross Section Storage – Portions of cross sections are often defined with ineffective flow areas, which represents water that has no conveyance. The extent of the storage within a cross section is an estimate. What if the ineffective flow areas were larger or smaller? How would this effect the results? This is another area that should be tested to see the sensitivity to the final solution.

Lateral Weir/Spillway Coefficients – Lateral weir/spillway coefficients can have a great impact on the results of a simulation, because they take water away or bring water into the main stream system. These coefficients can vary greatly for a lateral structure, depending upon their angle to the main flow, the velocity of the main flow, and other factors. The sensitivity of these coefficients should also be evaluated.

Bridge/Culvert Parameters – In general, bridge and culvert parameters normally only effect the locally computed water surface elevations just upstream and downstream of the structure. The effect that a bridge or culvert structure will have on the water surface is much greater in flat streams (a small increase in water surface can back upstream for a long distance if the river is flat). However, the sensitivity of the water surface elevations around a bridge or culvert may be very important to localized flooding. The bridge and culvert hydraulic parameters should also be evaluated to test their sensitivity.

CHAPTER 9

Viewing Results

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

Contents

- Cross Sections, Profiles, and Rating Curves
- Stage and Flow Hydrographs
- X-Y-Z Perspective Plots
- Tabular Output
- Viewing Results From The River System Schematic
- Viewing Ice Information
- Viewing Data Contained in an HEC-DSS File
- Exporting Results to HEC-DSS

Cross Sections, Profiles, and Rating Curves

Graphical displays are often the most effective method of presenting input data and computed results. Graphics allow the user to easily spot errors in the input data, as well as providing an overview of the results in a way that tables of numbers cannot.

Viewing Graphics on the Screen

To view a graphic on the screen, select **Cross Sections, Profiles,** or **Rating Curves** from the **View** menu on the HEC-RAS main window. Once you have selected one of these options, a window will appear with the graphic plotted in the viewing area. An example cross-section plot is shown in Figure 9.1. The user can plot any cross section by simply selecting the appropriate reach and river station from the list boxes at the top of the plot. The user can also step through the cross section plots by using the up and down arrows.

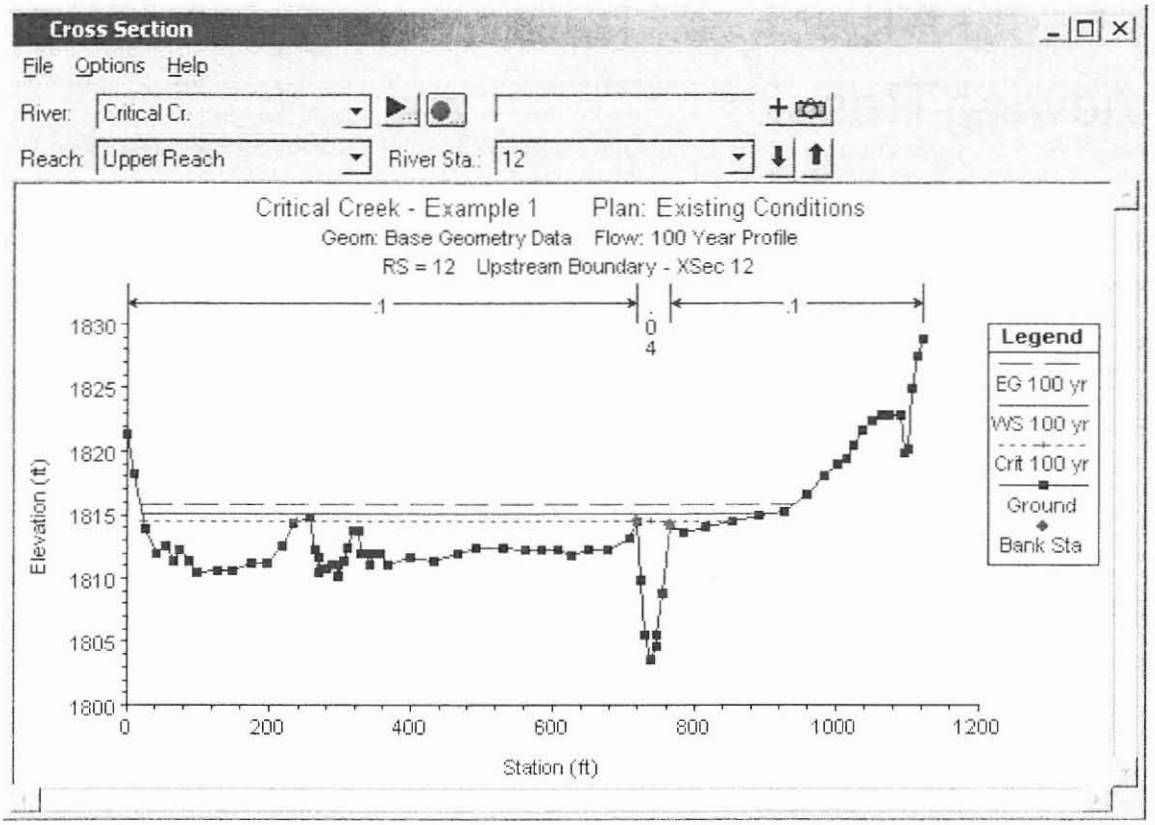


Figure 9.1 Example Cross Section Plot

An example profile plot is shown in Figure 9.2. The profile plot displays the water surface profile for the first reach in the river system. If there is more than one reach, additional reaches can be selected from the Options menu on or the reach button at the top of the window.

An example rating curve plot is shown in Figure 9.3. The rating curve is a plot of the water surface elevation versus flow rate for the profiles that were computed. A rating curve can be plotted at any location by selecting the appropriate reach and river station from the list boxes at the top of the plot.

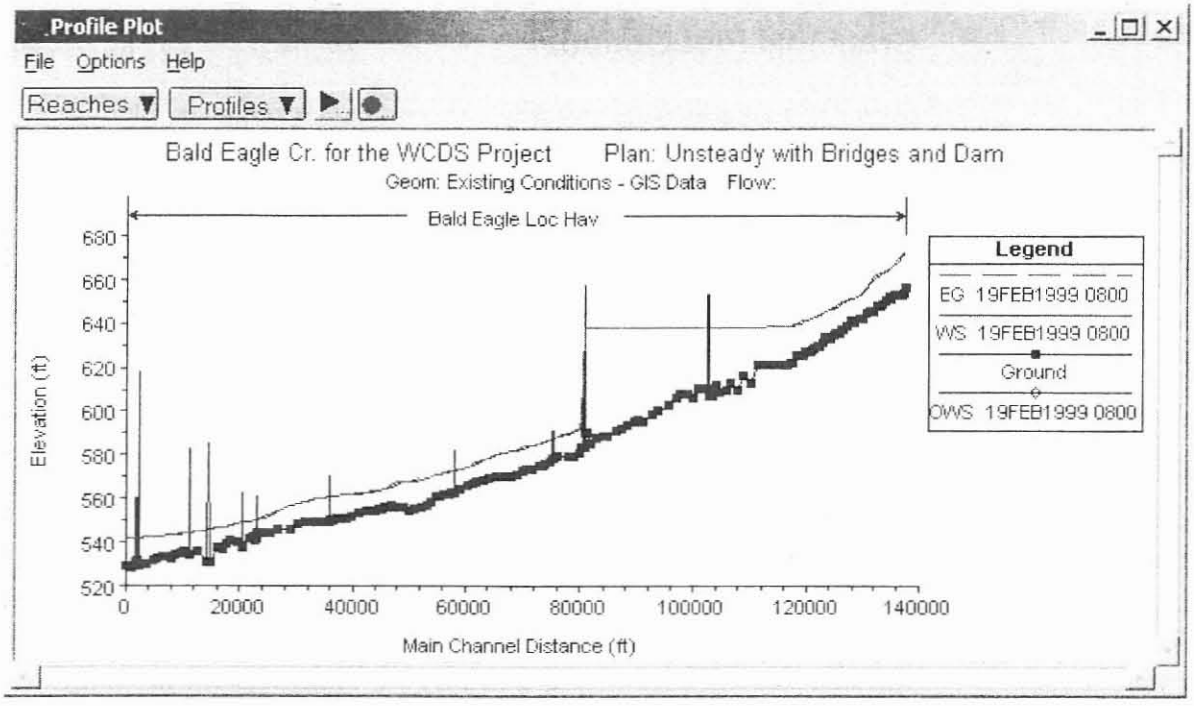


Figure 9.2 Example Profile Plot

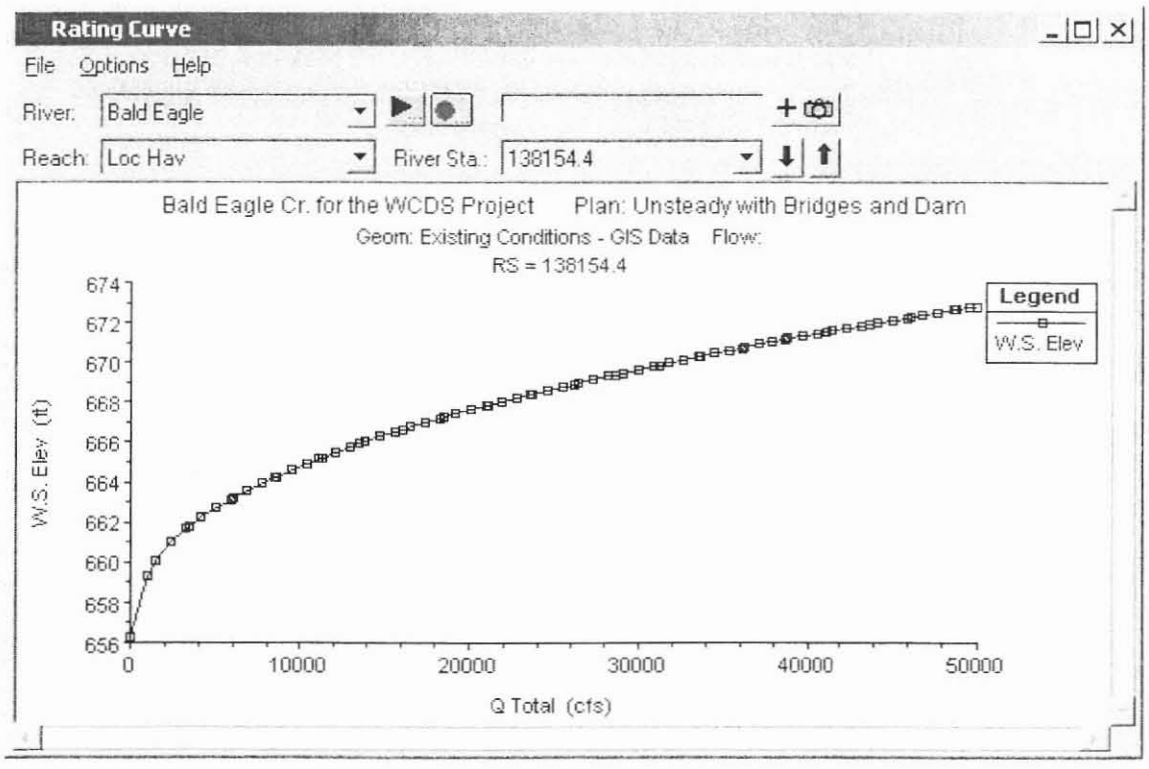


Figure 9.3 Example Rating Curve Plot

Graphical Plot Options

Several plotting features are available from the **Options** menu on all of the graphical plots. These options include: zoom in; zoom out; selecting which plans, profiles, reaches and variables to plot; and control over labels, lines, symbols, scaling, grid options, zoom window location, font sizes, and land marks. In addition to using the options menu at the top of each graphic window, if a user presses the right mouse button while the cursor is over a graphic, the options menu will appear right at the cursor location. In general, the options are about the same on all of the graphics.

Zoom In. This option allows the user to zoom in on a portion of the graphic. This is accomplished by selecting **Zoom In** from the **Options** menu, then specifying the area to zoom in on with the mouse. Defining the zoom area is accomplished by placing the mouse pointer at a corner of the desired zoom area. Then press down on the left mouse button and drag the mouse to define a box containing the desired zoom area. Finally, release the left mouse button and the viewing area will display the zoomed-in graphic. A small window showing the entire graphic will be placed in one of the corners of the graphic. This window is called the **Zoom Window**. The Zoom Window shows the entire graphic with a box around the zoomed in area. The user can move the zoom box or resize it in order to change the viewing area.

Zoom Out. This option doubles the size of the currently zoomed in graphic.

Full Plot. This option re-displays the graphic back into its original size before you zoomed in. Using the **Full Plot** option is accomplished by selecting **Full Plot** from the **Options** menu.

Pan. This option allows the user to move the graphic around while in a zoomed in mode. After zooming in, to move the graphic around, select **Pan** from the **Options** menu. Press and hold the left mouse button down over the graphic, then move the graphic in the desired direction.

Animate. This option was developed for unsteady flow output analysis, but can also be used for steady flow output. This option works with the cross section, profile, and X, Y, Z perspective plots. When this option is selected, a window will appear that allows the user to control the animation of any currently opened graphics. The user has the option to too "play" a graphic, which means to step through the time sequence of computed profiles. In a steady flow analysis, it can be used to switch between the profiles conveniently.

Plans. This option allows the user to select from the available Plans for plotting. The default plan is the currently opened plan. The user can select additional plans to view for comparison of results graphically.

Profiles. This option allows the user to select which profiles they would like to have displayed on the graphic. This option does not apply to the rating curve, it automatically plots all of the profiles.

Reaches. This option allows the user to select which river reaches they would like to have displayed. This option only applies to the profile plot.

Variables. This option allows the user to select whatever variables are available for plotting. The number and type of variables depends on what type of graphic is being displayed. The following is a list of variables that can be found on the profile plot: water surface, energy, critical water surface, observed water surfaces, reach labels, and left and right main channel bank stations, ice cover, left and right levees, pilot channels, sediment elevations, and lateral structures. The cross section plot is has the following eight variables: water surface, filled in water surface, energy, critical depth, observed water surface, ice cover, Manning's n values, and pilot channels.

Labels. This option allows the user to change the labels at the top of the plot. The user can select any or all of the following items to be added to the caption: project title, plan title, run date, run time, geometry title, flow title, river and reach names, cross section descriptions, cross section river stationing, and any user defined additional text.

Lines and Symbols. This option allows the user to change the line types, line colors, line widths, symbol types, symbol sizes, symbol colors, fill patterns, and the line labels. When the user selects this option, a window will appear as shown in Figure 9.4.

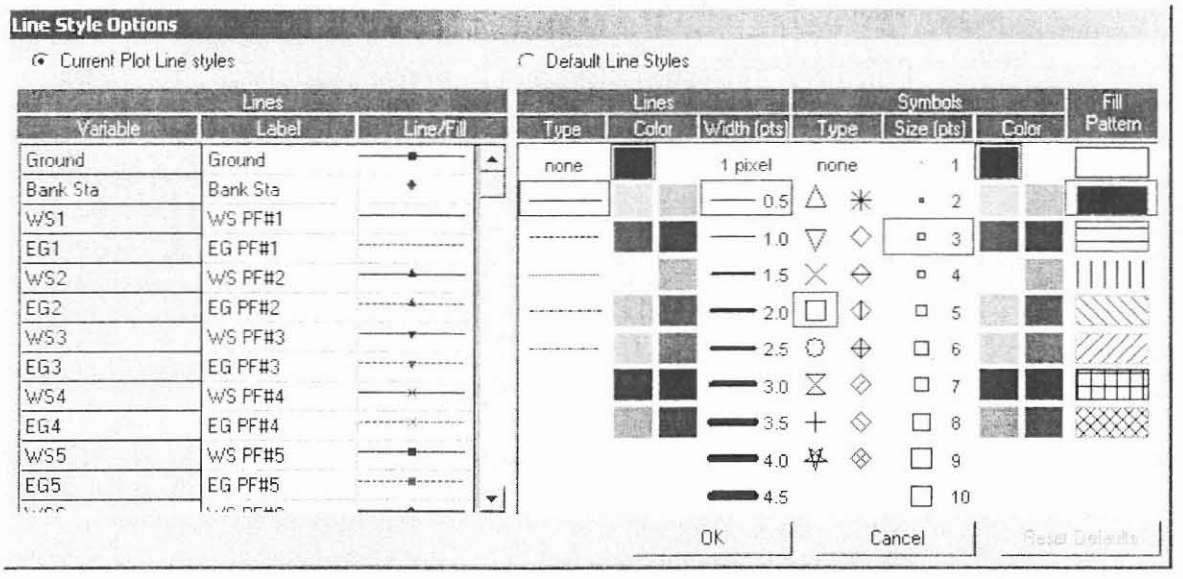


Figure 9.4 Line and Symbol Options Window

When the Line and Symbol Options window comes up, it will list only the information from the current plot. When this window is in the "Current Pict Line Styles" mode, the user can only change the information for the current plot. If the user wants to change the default line and symbol options for all of the plots, they must select **Default Line Styles** at the top of the window. When this option is selected, the user will be able to change the label, line, and symbol options for every variable that is plotted in the program. To use

this option, the user finds the variable that they want to change from the list on the left side of the window. Select that variable by clicking the left mouse button while over top of the variable. Once a variable is selected, the options that are set for that variable will be highlighted with a red box around each option. The user can change whatever option they want, as well as changing the label for that variable. If a variable does not have a default label, you cannot enter one for that variable. Once the user has made all of the changes that they want to all of the desired variables, they should press the **OK** button. The changes will be saved permanently, and any plot that is displayed within HEC-RAS will reflect the user-entered changes.

Scaling. This option allows the user to define the scaling used for the plot. Users are allowed to set the minimum, maximum, and labeling increment for the X and Y axis. Scaling can be set temporarily, or scaling can be set to be persistent (scaling stays constant for all cross sections). Persistent scaling is only available for the cross section and rating curve plots.

Grid. This option allows the user to overlay a grid on top of the graphic. Users have the option to have both major and minor tics displayed, as well as a border around the plot.

Zoom Window Location. This option allows the user to control which corner of the plot that the zoom window will be placed, and the size of the window.

Font Sizes. This option allows the user to control the size of all of the text displayed on the graphic.

Land Marks. This option is specific to profile plots. With this option the user can turn on additional labels that will be displayed as land marks below the invert of the channel. Two types of land marks can be displayed, cross section river stations or cross section descriptions. In addition to these two variables, once one of the two are displayed, the user can select to edit the land mark labels.

Plotting Velocity Distribution Output

The user has the option of plotting velocity distribution output from the cross section viewer. Velocity distributions can only be plotted at locations in which the user has specified that flow distribution output be calculated during the computations. To view the velocity distribution plot, first bring up a cross section plot (select "Cross Sections" from the view menu of the main HEC-RAS window). Next, select the cross section in which you would like to see the velocity distribution output. Select **Velocity Distribution** from the **Options** menu of the cross section window. This will bring up a pop up window (Figure 9.5) that will allow you to set the minimum velocity, maximum velocity, and velocity increment for plotting. In general, it is better to let the program use the maximum velocity range for plotting. Next, the user selects **Plot Velocity Distribution,** then press the "OK" button and the velocity distribution plot will appear as shown in Figure 9.6.

For details on how to select the locations for computing the velocity distribution, see Chapter 7 and 8 of the User's Manual. For information on how the velocity distribution is actually calculated, see Chapter 4 of the Hydraulic Reference Manual.

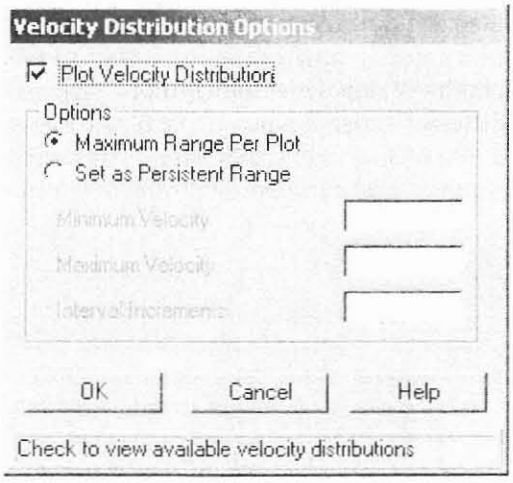


Figure 9.5 Velocity Distribution Options

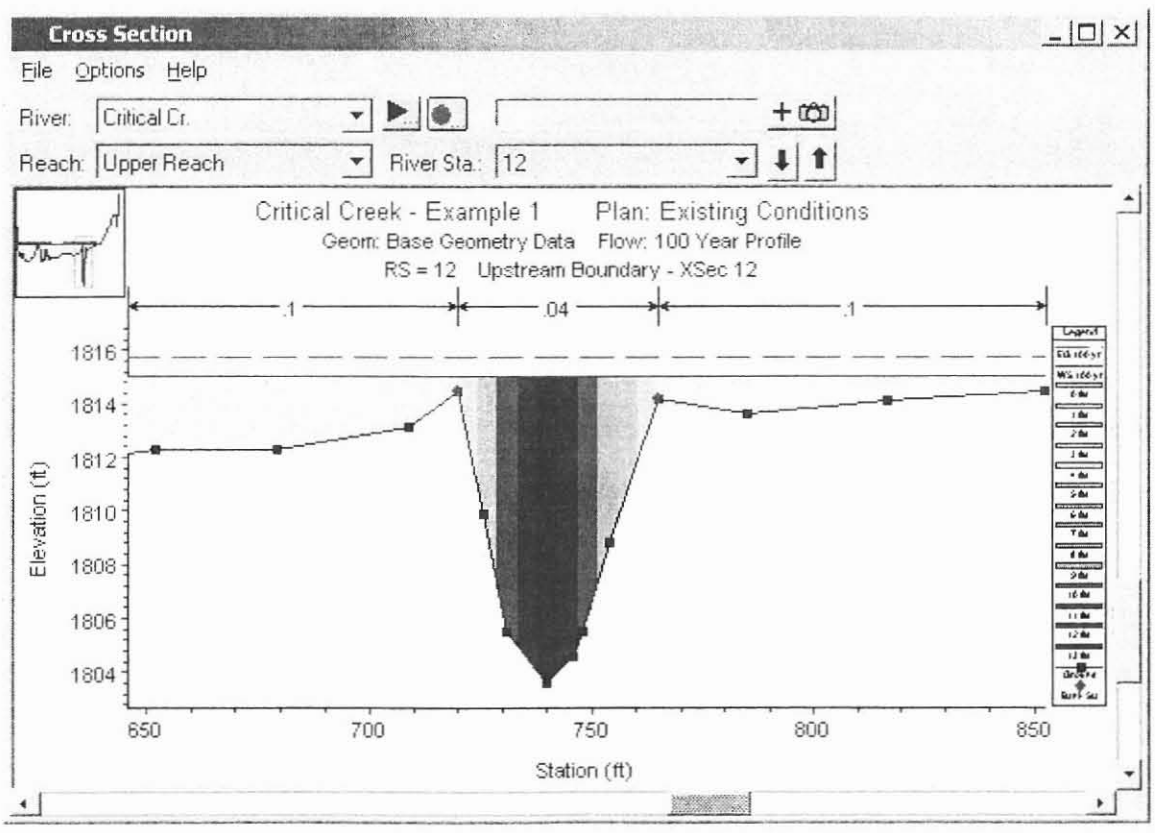


Figure 9.6 Velocity Distribution Plot

Plotting Other Variables in Profile

To plot variables other than the water surface in profile, select **General Profile** from the **View** menu of the main HEC-RAS window. Any variable that is computed at a cross section can be displayed in profile. An example would be to plot velocity versus distance. Other variables can be selected from the **Variables** option under the **Options** menu of the plot. The user can plot several different variable types at one time (e.g., velocity and area versus distance), but the scaling may not be appropriate when this is done. An example of plotting other variables in profile is shown in Figure 9.7.

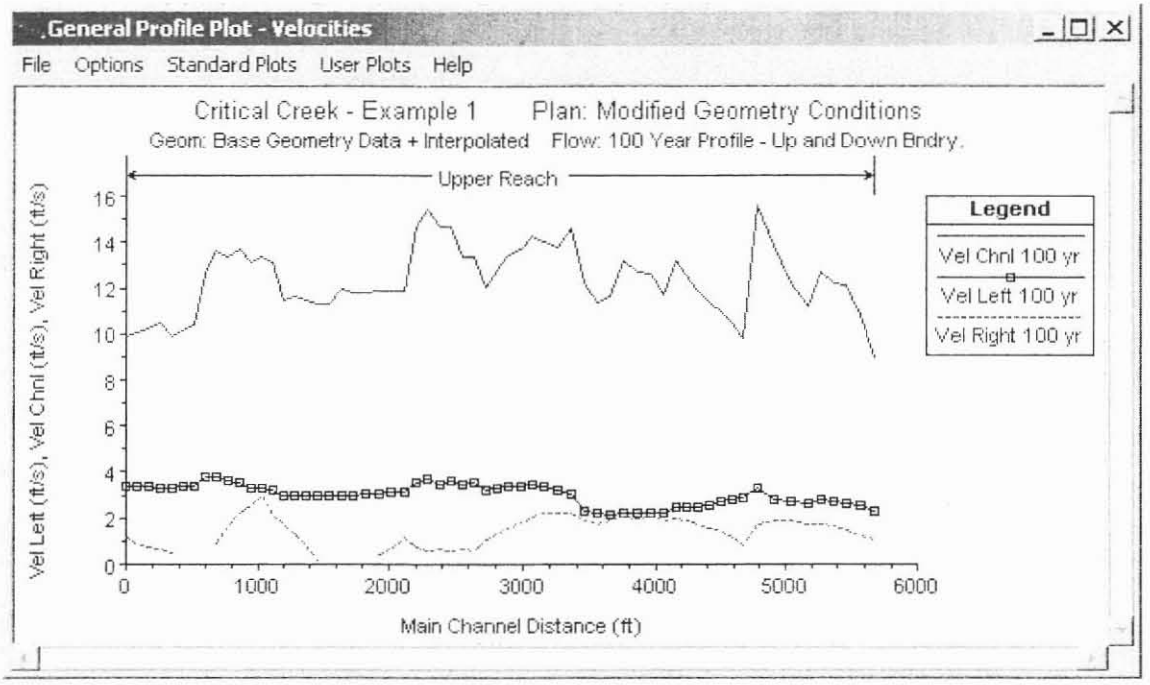


Figure 9.7 General Profile Plot of Variables Versus Distance

Plotting One Variable Versus Another

The rating curve plotting window has the ability to plot other variables besides discharge versus water surface elevation. Any variable that is computed at a cross section can be displayed against another computed variable (or variables). An example of this capability is shown in Figure 9.8. In this example, Discharge (x-axis) is being plotted against total flow area and main channel flow area (y-axis).

To plot other variables, the user selects the **X Axis Variable** and **Y Axis Variable** from the **Options** menu of the rating curve plotting window. When selected variables to plot, keep in mind that all variables selected for a particular axis should have a similar range in magnitude.

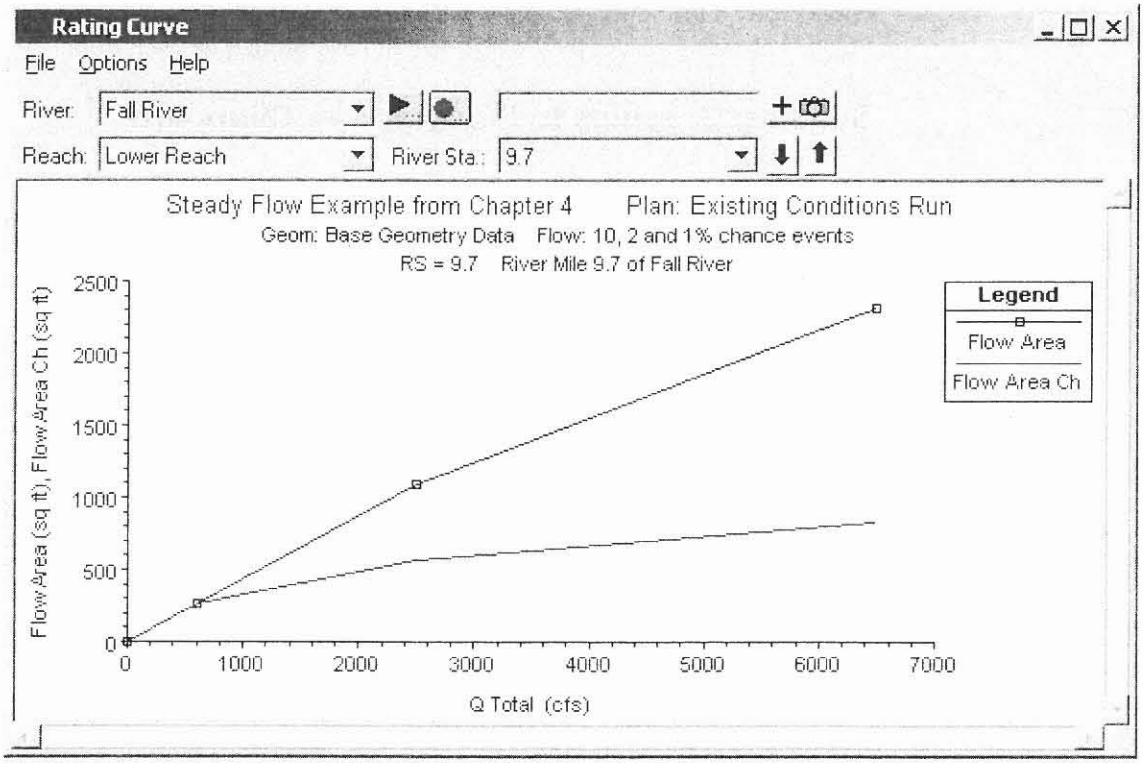


Figure 9.8 Example of Plotting One Variable Against Other Variables

Sending Graphics to the Printer or Plotter

All of the graphical plots in HEC-RAS can be sent directly to a printer or plotter. The printer or plotter used depends on what you currently have set as the default printer or plotter in the Windows Print Manager. To send a graphic to the printer or plotter, do the following:

1. Display the graphic of interest (cross section, profile, rating curve, X-Y-Z, or river system schematic) onto the screen.

2. Using the available graphics options (scaling, labels, grid, etc.), modify the plot to be exactly what you would like printed.

3. Select **Print Current** (or just **Print** on the profile plot) from the **File** menu of the displayed graphic. When this option is selected, a pop up window will appear allowing you to modify the default print options. Change any desired options and press the **Print** button. The graphic will be sent to the Windows Print Manager. The print manager will then send the plot to the default printer or plotter.

Note: The user can print multiple cross-sections at one time by using the **Print Multiple** option from the **File** Menu of the cross section and rating curve plots. This option also allows the user to establish how many cross sections or rating curves they would like to have printed on each page.

Sending Graphics to the Windows Clipboard

All of the HEC-RAS graphics can be sent to the Windows Clipboard. Passing a graphic to the clipboard allows that graphic to then be pasted into another piece of software (i.e., a word processor or another graphics program). To pass a graphic to the windows clipboard, and then to another program, do the following:

1. Display the graphic of interest on the screen.

2. Using the options menu, modify the plot to be exactly what you want.

3. Select **Copy to Clipboard** from the **File** menu of the displayed graphic. The plot will automatically be sent to the Windows Clipboard.
4. Bring up the program that you want to paste the graphic into. Select **Paste** from the **Edit** menu of the receiving program. Once the graphic is pasted in, it can be re-sized to the desired dimensions.

HEC-RAS sends and displays all graphics in a Window's Meta file format. Since Meta files are vector based graphics, the graphic can be resized without causing the image to distort.

Stage and Flow Hydrographs

If the user has performed an unsteady flow analysis, then stage and flow hydrographs will be available for viewing. To view a stage and/or flow hydrograph, the user selects **Stage and Flow Hydrographs** from the **View** menu of the main HEC-RAS window. When this option is selected a plot will appear as shown in Figure 9.9. The user has the option to plot just the stage hydrograph, just the flow hydrograph, or both as shown in the figure. Additionally, there are three tabs on the plot. The tabs are for plotting (**Plot**), viewing the data in tabular form(**Table**), and plotting a rating curve of the event (**Rating Curve**). By default the window comes up in a plotting mode.

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

There are several options available for viewing this graphic. These options are the same as described previously for the cross section, profile, and rating curve plots. Additionally, this graphic can be sent to the windows clipboard, or the printer, as described under the previous plots.

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

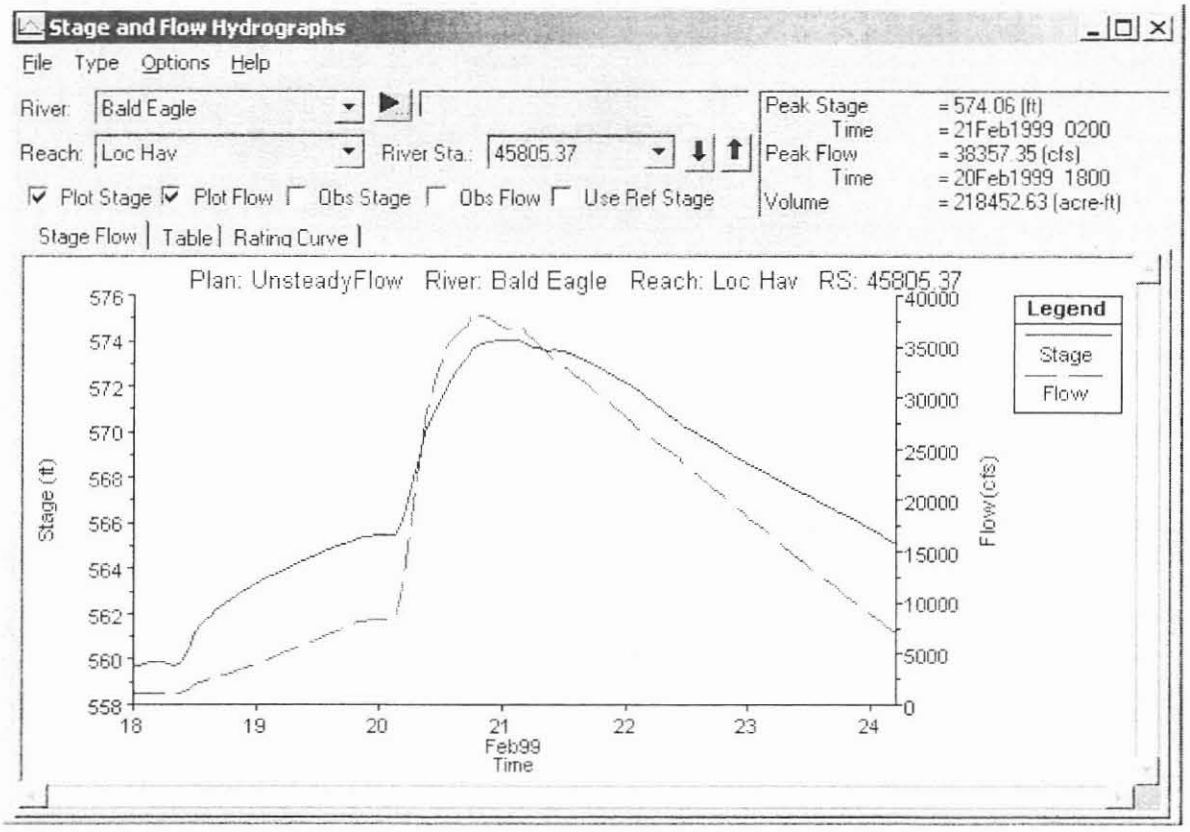


Figure 9.9 Stage and Flow Hydrograph Plot

X-Y-Z Perspective Plots

Another type of graphic available to the user is the X-Y-Z Perspective Plot. The X-Y-Z plot is a 3-dimensional plot of multiple cross sections within a reach. An example X-Y-Z Perspective plot is shown in Figure 9.10.

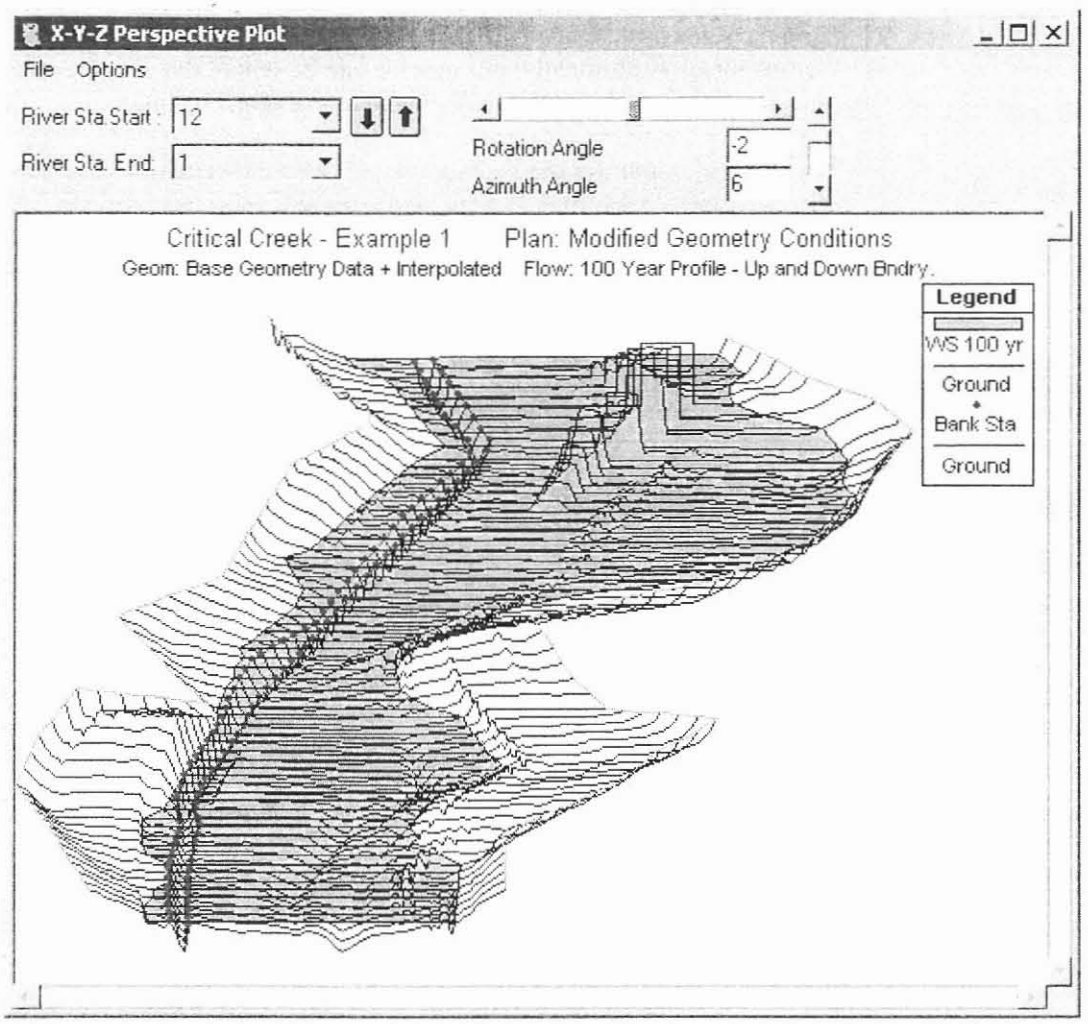


Figure 9.9 Example X-Y-Z Perspective Plot

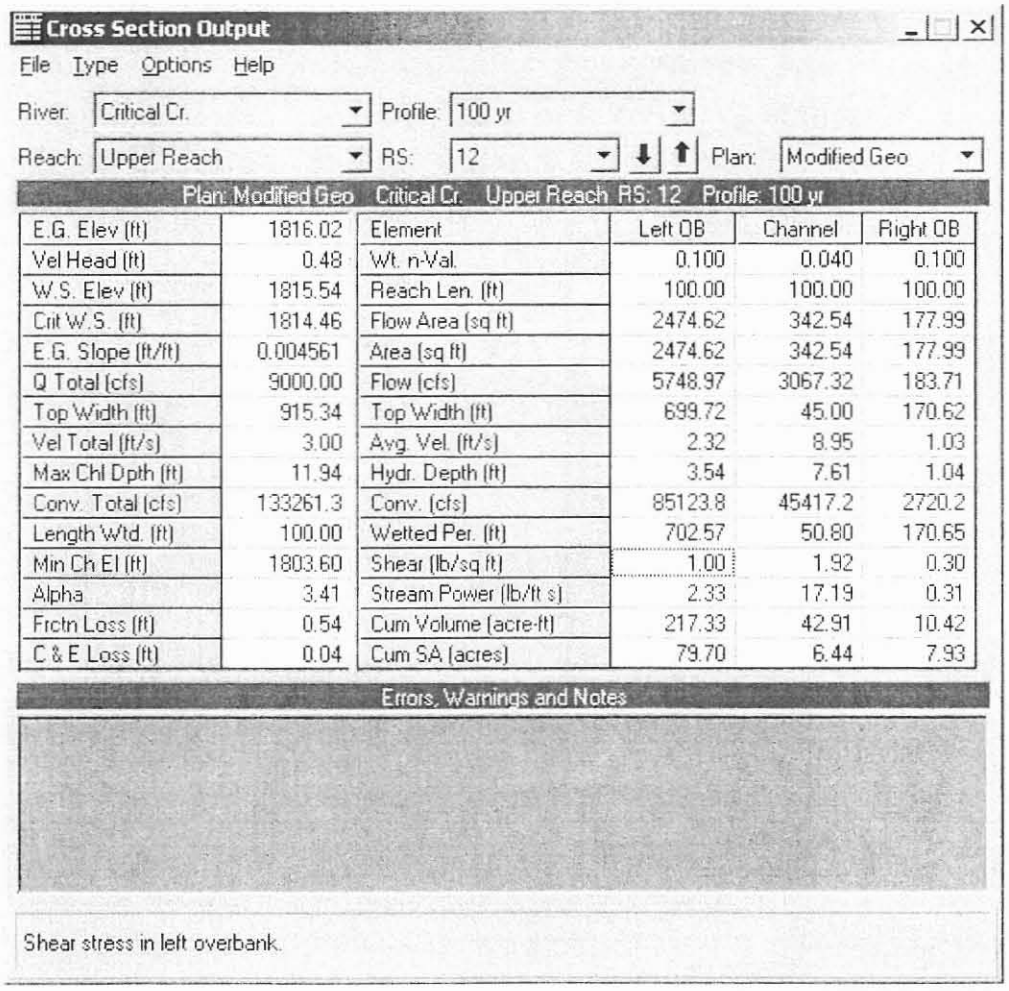
The user has the ability to select which reaches to be plotted, the range of the river stations, and which plans and profiles to be displayed. The plot can be rotated left and right, as well as up and down, in order to get different perspectives of the river system. Zoom in and zoom out features are available, as well as the ability to move around with scroll bars. The user can choose to overlay the water surface or not. The user has the ability to overlay a grid on the plot, as well as a legend and labels at the top. The graphic can be sent to the printer/plotter or the clipboard just like any other plot. Sending the graphic to the printer or clipboard is accomplished by selecting the **Print** or **Clipboard** options from the **File** menu. The user also has the option to reverse the order in which the water surface profiles are displayed. This option allows the user to display the higher water surfaces first, such that the lower profiles are not covered up.

Tabular Output

Summary tables of the detailed water surface profile computations are often necessary to analyze and document simulation results. Tabular output allows the user to display large amounts of detailed information in a concise format. HEC-RAS has two basic types of tabular output, detailed output tables and profile summary tables.

Detailed Output Tables

Detailed output tables show hydraulic information at a single location, for a single profile. To display a detailed output table on the screen, select **Detailed Output Table** from the **View** menu of the main HEC-RAS window. An example detailed output table is shown in Figure 9.10.



Cross Section Output

File Type Options Help

River: Critical Cr. Profile: 100 yr

Reach: Upper Reach RS: 12 Plan: Modified Geo

Plan: Modified Geo Critical Cr. Upper Reach RS: 12 Profile: 100 yr

E.G. Elev (ft)	1816.02	Element	Left OB	Channel	Right OB
Vel Head (ft)	0.48	Wt. n-Val.	0.100	0.040	0.100
W.S. Elev (ft)	1815.54	Reach Len. (ft)	100.00	100.00	100.00
Crit W.S. (ft)	1814.46	Flow Area (sq ft)	2474.62	342.54	177.99
E.G. Slope (ft/ft)	0.004561	Area (sq ft)	2474.62	342.54	177.99
Q Total (cfs)	9000.00	Flow (cfs)	5748.97	3067.32	183.71
Top Width (ft)	915.34	Top Width (ft)	699.72	45.00	170.62
Vel Total (ft/s)	3.00	Avg. Vel. (ft/s)	2.32	8.95	1.03
Max Chl Dpth (ft)	11.94	Hydr. Depth (ft)	3.54	7.61	1.04
Conv. Total (cfs)	133261.3	Conv. (cfs)	85123.8	45417.2	2720.2
Length Wtd. (ft)	100.00	Wetted Per. (ft)	702.57	50.80	170.65
Min Ch El (ft)	1803.60	Shear (lb/sq ft)	1.00	1.92	0.30
Alpha	3.41	Stream Power (lb/ft s)	2.33	17.19	0.31
Frctn Loss (ft)	0.54	Cum Volume (acre-ft)	217.33	42.91	10.42
C & E Loss (ft)	0.04	Cum SA (acres)	79.70	6.44	7.93

Errors, Warnings and Notes

Shear stress in left overbank.

Figure 9.10 Example Cross Section Detailed Output Table

By default, this table comes up displaying detailed output for cross sections. Any cross section can be displayed in the table by selecting the appropriate

river, reach and river station from the list boxes at the top of the table. Also, any of the computed profiles can be displayed by selecting the desired profile from the profile list box. Additionally, different plans can be viewed by selecting a plan from the plan list box.

Users can also view detailed hydraulic information for other types of nodes. Other table types are selected from the **Type** menu on the detailed output table window. The following types are available in addition to the normal cross section table (which is the default):

Culvert. The culvert table type brings up detailed culvert information. This table can be selected for normal culverts, or for culverts that are part of a multiple opening river crossing. An example culvert specific table is shown in Figure 9.12.

Culvert Output

File Type Options Help

River: Spring Creek Profile: 50 yr Culv Group: Culvert # 1

Reach: Culvrt Reach RS: 20.237 Plan: Base Plan

Plan: Base Plan Spring Creek Culvrt Reach RS: 20.237 Culv Group: Culvert # 1 Profile: 50 yr

Culv Q (cfs)	972.24	Culv Full Len (ft)	50.00
# Barrels	1	Culv Vel US (ft/s)	6.98
Q Barrel (cfs)	972.24	Culv Vel DS (ft/s)	6.98
E.G. US. (ft)	34.10	Culv Inv El Up (ft)	25.10
W.S. US. (ft)	34.03	Culv Inv El Dn (ft)	25.00
E.G. DS (ft)	33.06	Culv Frctn Ls (ft)	0.06
W.S. DS (ft)	32.90	Culv Exit Loss (ft)	0.59
Delta EG (ft)	1.03	Culv Entr Loss (ft)	0.38
Delta WS (ft)	1.13	Q Weir (cfs)	27.76
E.G. IC (ft)	30.85	Weir Sta Lft (ft)	959.77
E.G. OC (ft)	34.10	Weir Sta Rgt (ft)	1040.61
Culvert Control	Outlet	Weir Submerg	0.00
Culv WS Inlet (ft)	31.10	Weir Max Depth (ft)	0.40
Culv WS Outlet (ft)	31.00	Weir Avg Depth (ft)	0.25
Culv Nml Depth (ft)		Wr Flw Area (sq ft)	20.40
Culv Crt Depth (ft)	3.32	Min El Weir Flow (ft)	33.71

Errors, Warnings and Notes

During the culvert inlet computations, the program could not balance the culvert/weir flow. The reported inlet energy grade answer may not be valid.

Flow over the weir.

Figure 9.12 Example Culvert Type of Cross Section Table

Bridge. The bridge table type brings up detailed output for the cross sections inside the bridge as well as just upstream of the bridge. The bridge table type

can be selected for normal bridge crossings, or for bridges that are part of a multiple opening river crossing. An example of the bridge specific cross section table is shown in Figure 9.13.

Bridge Output

File Type Options Help

River: Butte Creek Profile: 50 yr Opening Single BR

Reach Tributary Riv Sta: 0.22

Plan: plan2 Butte Creek Tributary RS: 0.22 Profile: 50 yr Opening: Single BR

E.G. US. (ft)	82.89	Element	Inside BR US	Inside BR DS
W.S. US. (ft)	82.76	E.G. Elev (ft)	82.89	82.87
Q Total (cfs)	500.00	W.S. Elev (ft)	82.76	82.58
Q Bridge (cfs)	337.54	Crit W.S. (ft)	76.73	76.73
Q Weir (cfs)	162.46	Max Chl Dpth (ft)	12.76	12.58
Weir Sta Lft (ft)	218.89	Vel Total (ft/s)	4.12	4.65
Weir Sta Rgt (ft)	298.64	Flow Area (sq ft)	121.28	107.48
Weir Submerg	0.27	Froude # Chl	0.08	0.09
Weir Max Depth (ft)	0.89	Specif Force (cu ft)	577.68	566.92
Min Weir El (ft)	82.01	Hydr Depth (ft)	1.55	1.42
Min El Prs (ft)	79.00	W.P. Total (ft)	130.85	128.31
Delta EG (ft)	0.47	Conv. Total (cfs)		
Delta WS (ft)	0.51	Top Width (ft)	78.01	75.54
BR Open Area (sq ft)	65.66	Frctn Loss (ft)		
BR Open Vel (ft/s)	5.14	C & E Loss (ft)		
Coef of Q		Shear Total (lb/sq ft)		
Br Sel Method	Press/Weir	Power Total (lb/ft s)		

Errors, Warnings and Notes

Note: Yarnell answer is not valid if the water surface is above the low chord or if there is weir flow. The Yarnell answer has been disregarded.

Note: Momentum answer is not valid if the water surface is above the low chord or if there is weir flow. The momentum answer has been disregarded.

Note: The downstream water surface is above the minimum elevation for pressure flow. The orifice equations were used for pressure flow.

Energy grade elevation at bridge or culvert (final answer).

Figure 9.13 Example Bridge Type of Cross Section Table

Multiple Opening. The multiple opening type of table is a combination of the cross section table and the bridge and culvert tables. That is, if the user has defined multiple opening (bridges, culverts, and conveyance areas), then this table can be used to view the hydraulic results for each specific opening.

Inline Structure. The Inline Structure type of table can be used to view detailed output for any inline weirs and/or gated spillways that have been entered by the user.

Lateral Structure. The Lateral Structure type of table can be used for viewing detailed output from a lateral weir, gated spillway, culvert, and rating curves.

Storage Area. This table provides output about an individual storage area. Information includes water surface elevation, total inflow, total outflow, and net inflow.

Storage Area Connection. This table provides detailed information about storage area connections. Storage area connections can consist of weirs, gated spillways, and culverts.

Pump Stations. This table provides detailed information about pump stations. Pump station output includes to and from water surface elevations, total flow, flow through each pump group, flow through each pump, head difference, and efficiency.

Flow Distribution In Cross Sections. The Flow Distribution table type can be used to view the computed flow distribution output at any cross section where this type of output was requested. An example of the flow distribution table output is shown in Figure 9.14.

Flow Distribution Output

File Type Options Help

River: Critical Cr. Profile: 100 yr

Reach Upper Reach Riv Sta: 12

Plan: Modified Geo Critical Cr. Upper Reach RS: 12 Profile: 100 yr

Left Sta (ft)	Right Sta (ft)	Flow (cfs)	Area (sq ft)	W.P. (ft)	% Conv.	Hydr D. (ft)	Velocity (ft/s)
0.00	72.00	298.90	150.01	52.17	3.32	2.90	1.99
72.00	144.00	912.27	333.48	72.08	10.14	4.63	2.74
144.00	216.00	804.58	309.20	72.05	8.94	4.29	2.60
216.00	288.00	326.78	181.00	72.97	3.63	2.51	1.81
288.00	360.00	585.69	257.11	73.14	6.51	3.57	2.28
360.00	432.00	755.81	297.79	72.03	8.40	4.14	2.54
432.00	504.00	603.51	260.15	72.01	6.71	3.61	2.32
504.00	576.00	498.87	232.05	72.00	5.54	3.22	2.15
576.00	648.00	557.22	247.99	72.01	6.19	3.44	2.25
648.00	720.00	404.78	204.82	72.10	4.50	2.84	1.98
LB 720.00	724.50	46.34	12.43	5.67	0.51	2.76	3.73
724.50	729.00	179.78	28.47	5.89	2.00	6.33	6.31
729.00	733.50	403.52	44.07	5.22	4.48	9.79	9.16

Errors, Warnings and Notes

Flow in subsection defined by left and right stations

Figure 9.14 Example of the Flow Distribution Type of Table

At the bottom of each of the detailed output tables are two text boxes for displaying messages. The bottom text box is used to display the definition of the variables listed in the table. When the user presses the left mouse button over any data field, the description for that field is displayed in the bottom text box. The other text box is used to display any **Errors, Warnings, and Notes** that may have occurred during the computations for the displayed cross section.

Detailed Output Table Options

Plans. This option allows the user to select which plan, and therefore output file, they would like to view. This option is available from a list box at the upper right hand side of the window.

Under the **Options** menu of the cross section table window, the user has the following options:

Include Interpolated XS's. This option allows the user to either view interpolated cross-section output or not. Turning the "include interpolated XS's" option on (which is the default), allows interpolated sections to be selected from the river station box. Turning this option off gets rid of all the interpolated sections from the river station selection box, and only the user entered cross-sections are displayed.

Include Errors, Warnings, and Notes in Printout. This option allows the user to have the errors, warnings, and notes information printed below the table, when the option to print the table is selected.

Units System For Viewing. This option allows the user to view the output in either English or Metric units. It does not matter whether the input data is in English or Metric, the output can be viewed in either system.

Profile Summary Tables

Profile summary tables are used to show a limited number of hydraulic variables for several cross sections. To display a profile summary table on the screen, select **Profile Summary Table** from the **View** menu of the main HEC-RAS window. An example profile summary table is shown in Figure 9.15.

Profile Output Table - Standard Table 1

File Options Std. Tables Help

HEC-RAS Plan: Modified Geo River: Critical Cr. Reach: Upper Reach Profile: 100 yr Reload Data

Reach	River Sta	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)	
Upper Reach	12	9000.00	1803.60	1815.54	1814.46	1816.02	0.004567	8.96	2993.81	915.30	0.57
Upper Reach	11.8*	9000.00	1803.02	1814.58	1814.14	1815.44	0.006443	10.78	2403.45	817.51	0.69
Upper Reach	11.6*	9000.00	1802.44	1813.40	1813.40	1814.67	0.008386	12.14	2043.65	764.50	0.80
Upper Reach	11.4*	9000.00	1801.86	1812.43	1812.47	1813.82	0.008533	12.25	1894.08	698.93	0.81
Upper Reach	11.2*	9000.00	1801.28	1811.29	1811.46	1812.89	0.009621	12.73	1701.14	646.20	0.86
Upper Reach	11	9000.00	1800.70	1810.68	1810.42	1811.90	0.007043	11.20	1888.44	651.48	0.75
Upper Reach	10.75*	9000.00	1799.13	1809.51	1809.51	1810.91	0.008530	12.22	1854.70	699.37	0.82
Upper Reach	10.5*	9000.00	1797.55	1807.93	1808.34	1809.61	0.012265	13.89	1866.64	830.45	0.96
Upper Reach	10.25*	9000.00	1795.97	1805.90	1806.41	1807.69	0.019512	15.59	1795.08	857.69	1.16

Total flow in cross section

Figure 9.15 Example Profile Table

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

Standard Table 1. The is the default profile type of table. This table gives you a summary of some of the key output variables.

Standard Table 2. This is the second of the standard summary tables. This table provides information on the distribution of flow between the left overbank, main channel, and right overbank. This table also shows the friction losses, as well as contraction and expansion losses that occurred between each section. Energy losses displayed at a particular cross section are for the losses that occurred between that section and the next section downstream.

Four XS Culvert. This standard table provides summary results for the four cross sections around each of the culverts in the model. The four cross sections are the two immediately downstream and the two immediately upstream of the culvert. This table will list all of the culverts in the model for the selected reaches.

Culvert Only. This standard table provides hydraulic information about the culvert, as well as the inlet control and outlet control computations that were performed.

Six XS Bridge. This table provides summary results for the six cross sections that make up the transition of flow around a bridge. The six cross sections include the two cross sections just downstream of the bridge; the two cross sections inside of the bridge; and the two cross sections just upstream of the bridge. The program will display results for all the bridges in the model within the selected reaches. When viewing this table, on occasion there will be no displayed results for the cross sections inside of the bridge. This occurs

only when the user has selected a bridge modeling approach that does not compute results inside of the bridge. This includes: Yarnell's method; both pressure flow equations; and pressure and weir flow solutions.

Bridge Only. The bridge only table shows summary information specifically for bridges.

Bridge Comparison. The bridge comparison table shows the results for all of the user selected bridge modeling approaches that were computed during the computations. For example, the program can calculate low flow bridge hydraulics by four different methods. The resulting upstream energy for the user selected methods will be displayed in this table.

Multiple Opening. This table shows a limited number of output variables for each opening of a multiple opening river crossing.

Four XS Inline Structure. This table displays summary results of the four cross sections immediately around an inline weir and/or gated spillway. The four cross sections are the two immediately upstream and the two immediately downstream of the inline weir and/or gated spillway.

Inline Structure Only. This table shows the final computed water surface and energy just upstream of each of the inline weir and/or gated spillways. In addition to these elevations, the table displays the total flow, the flow over the weir, and the total flow through all of the gates.

Lateral Structure. This table shows a limited set of output variables for all of the lateral weir/spillway structures within the selected reaches.

Encroachment 1, 2, and 3. These three standard tables provide various types of output for the computations of floodway encroachments.

HEC-FDA. This table provides information that can be exported to the HEC Flood Damage Analysis (FDA) program. The table displays total flow, channel invert elevation, and water surface elevation.

HEC-5Q. This table provides information that can be exported to the HEC-5Q (river and reservoir water quality analysis) program. The table displays only the specific parameters required by the HEC-5Q program.

Ice Cover. This table shows summary output of ice information. This table was designed for performing a study that includes ice cover.

Junctions. This summary table provides a limited set of output for all of the cross sections that bound a junction. This table will show this output for all of the junctions found in the model.

Storage Areas. This table shows a limited amount of output for all of the storage areas in the model. Output includes: water surface elevation; minimum storage area elevation; surface area; and volume.

Pump Stations. This table shows a limited amount of output for any of the pump stations contained within the model.

To view one of the types of tables, select the desired table type from the **Std. Tables** menu on the profile summary table. In addition to the various types of profile tables, the user can specify which plans, profiles and reaches to include in the table. The plans, profiles and reaches options are available from the **Options** menu on the profile plot.

The user also has the ability to turn the viewing of interpolated cross sections on or off. The default is to view all cross-sections, including the interpolated ones. To prevent the interpolated sections from showing up in the table, de-select **Include Interpolated XS's** from the **Options** menu.

Another feature available to users is the ability to set the number of decimal places that will be displayed for any variable of the pre-defined tables. Once a pre-defined table is selected from the **Tables** menu, select **Standard Table # Dec Places** from the **Options** menu. A window will appear displaying the current number of decimal places for each variable. The user can change the number of decimal places to what ever they wish.

User's also have the ability to view profile output tables in either English or metric units. This is available from the **Options** menu on the profile tables. It does not matter whether the input data is in English or metric, the output can be viewed in either system.

User Defined Output Tables

A special feature of the profile summary tables is the ability for users to define their own output tables. User defined output tables are available by selecting **Define Table** from the **Options** menu of the profile table. When this option is selected, a window will appear, as shown in Figure 9.16. At the top of the window is a table for the user selected variable headings (Table Column Headings), the units, and the number of decimal places to be displayed for each variable. Below this table is a table containing all of the available variables that can be included in your user-defined table. The variables are listed in alphabetical order. Below the list of variables is a message box that is used to display the definition of the selected variable.

To get a definition of a particular variable, simply click the left mouse button once while the mouse pointer is over the desired variable. The description of the variable will show up at the bottom of the window. To add variables to the column headings, simply double click the left mouse button while the mouse pointer is over the desired variable. The variable will be placed in the active field of the table column headings. To select a specific column to place a variable in, click the left mouse button once while the mouse pointer is over the desired table column field. To delete a variable from the table headings, double click the left mouse button while the mouse pointer is over the variable that you want to delete. The number of decimal places for each variable can be changed by simply typing in a new value.

User defined tables are limited to 15 variables. Once you have selected all of the variables that you want, press the **OK** button at the bottom of the window. The profile table will automatically be updated to display the new table.

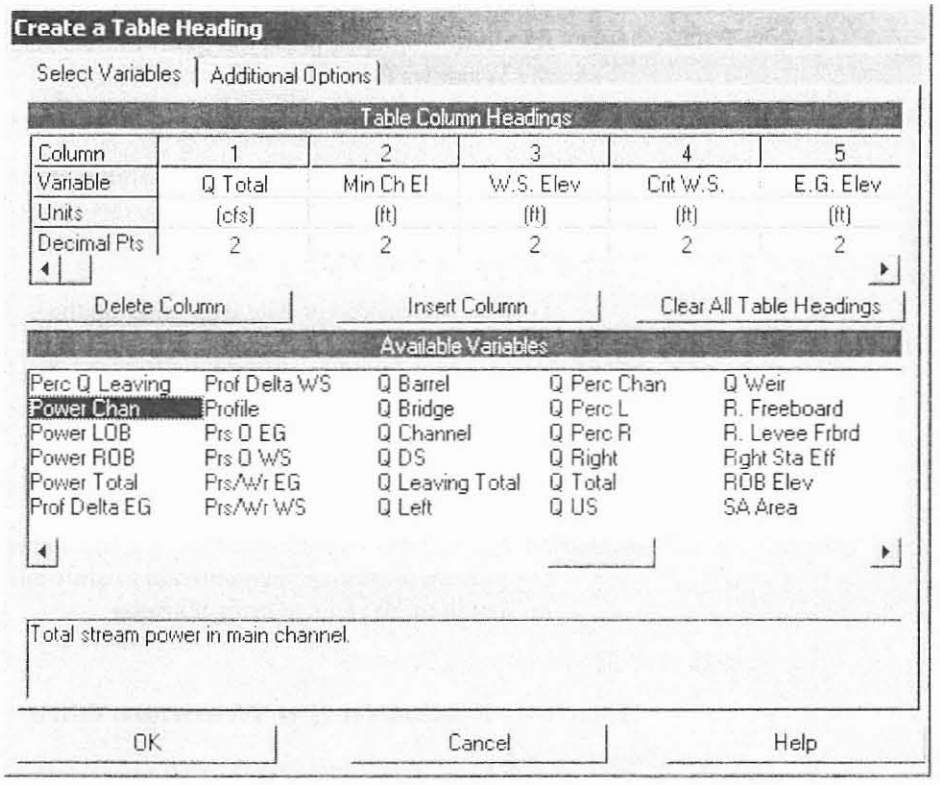


Figure 9.16 User Defined Tables Window

Once you have the table displayed in the profile table window, you can save the table headings for future use. To save a table heading, select **Save Table** from the **Options** menu on the profile table window. When this option is selected, a pop up window will appear, prompting you to enter a name for the table. Once you enter the name, press the **OK** button at the bottom of the pop up window. The table name will then be added to a list of tables included under the **User Tables** menu on the profile table window. To delete a table from the list of user defined tables, select **Remove Table** from the **Options** menu of the profile table window. When this option is selected, a pop up window will appear displaying a list of all the user-defined tables. Click the left mouse button over the tables that you want to delete, then press the **OK** button. The selected tables will then be deleted from the **User Tables** menu list.

Sending Tables to the Printer

To send a table to the printer, do the following:

1. Bring up the desired table from the tabular output (cross section or profile tables) section of the program.

2. Select **Print** from the **File** menu of the displayed table. When this option is selected, a pop up window will appear allowing you to modify the default print options. Once you have set the printer with the desired options, press the **Print** button. The table will be sent to the Windows Print Manager. The Windows Print Manager will control the printing of the table.

The profile summary type of tables, allow you to print a specific portion of the table, rather than the entire table. If you desire to only print a portion of the table, do the following:

1. Display the desired profile type table on the screen.

2. Using the mouse, press down on the left mouse button and highlight the area of the table that you would like to print. To get an entire row or column, press down on the left mouse button while moving the pointer across the desired row or column headings.

3. Select **Printer** from the **File** menu of the displayed table. Only the highlighted portion of the table and the row and column headings will be sent to the Windows Print Manager.

Sending Tables to the Windows Clipboard

To pass a table to the Windows Clipboard, and then to another program, do the following:

1. Display the desired table on the screen.

2. Select **Copy to Clipboard** from the **File** menu of the displayed table.

3. Bring up the program that you want to pass the table into. Select **Paste** from the **Edit** menu of the receiving program.

Portions of the profile tables can be sent to the Clipboard in the same manner as sending them to the printer.

Viewing Results From the River System Schematic

The user has the option of either bringing up graphics and tables from the **View** menu on the main HEC-RAS window (as discussed above), or from the river system schematic (found under geometric data). Once data have been entered, and a successful simulation has been made, the user can interact with the river system schematic. When the left mouse button is pressed over the river system schematic, a pop up menu will appear listing options that are relevant to the area of the schematic that is located under the mouse pointer. An example of this is shown in Figure 9.17.

In Figure 9.17, the pop up menu shown comes up whenever the user presses the left mouse button over a cross section. In this particular example, the mouse button was pressed over the cross section located at river station 10.0 of the Upper reach of Fall river. As shown in the menu, the user has the choice of editing the cross section data; plotting the cross section; plotting the profile for the reach containing this cross section; bringing up the XYZ plot for that reach; viewing tabular output; plotting the computed rating curve at this cross section; or viewing a picture of the location. Other pop up menus are available for bridges; culverts; junctions; and reach data.

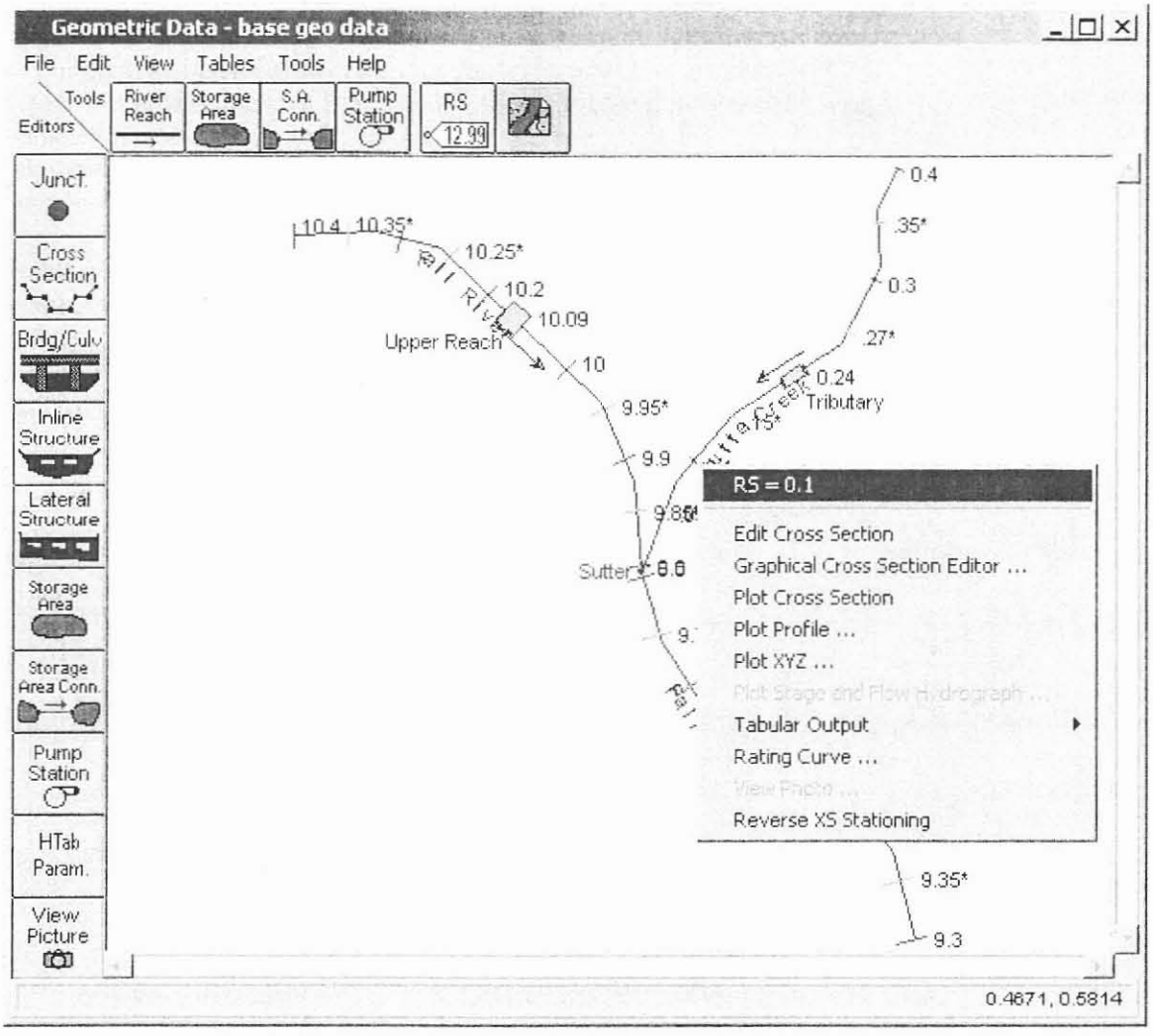


Figure 9.17 Geometric Data Window With Pop up Menu

Viewing Ice Information

River ice information can be viewed both in a graphical and tabular format.

Viewing Graphical Ice Information on the Screen

To view graphical ice information on the screen, select either **Cross Sections**, **Profiles**, or **X-Y-Z Perspective Plot** from the View menu on the HEC-RAS main window.

Cross Section Plot. Figure 9.18 is an example cross section plot displaying ice. The ice cover is displayed by selecting **Variables** under the **Options** menu, then selecting the **Ice Cover** option. The ice thicknesses in the right overbank, main channel, and left overbank are displayed. The default color and fill pattern can be changed by the user by selecting **Lines and Symbols** under the **Options** menu. Note that multiple profiles and multiple plans can be displayed on the same plot.

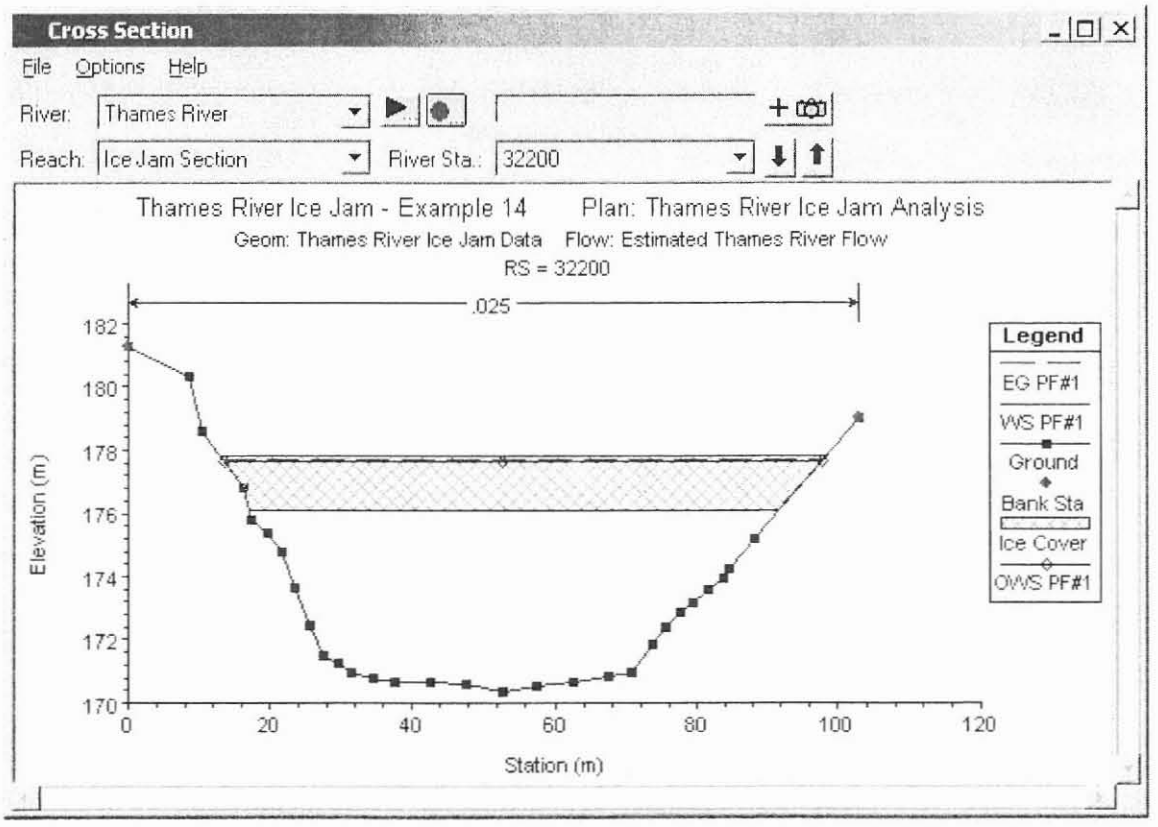


Figure 9.18 Cross section Plot with ice

Profiles Plot. An example of a profile plot with ice is shown in Figure 9.19. In this case, the **WS-EG Profile** was selected. As with the Cross Section plot, the ice cover is displayed by selecting **Variables** under the **Options** menu,

then selecting the **Ice Cover** option. The ice thicknesses in the right overbank, main channel, and left overbank are displayed. The default color and fill pattern can be changed by the user by selecting **Lines and Symbols** under the Options menu. Note that multiple profiles and multiple plans can be displayed on the same plot.

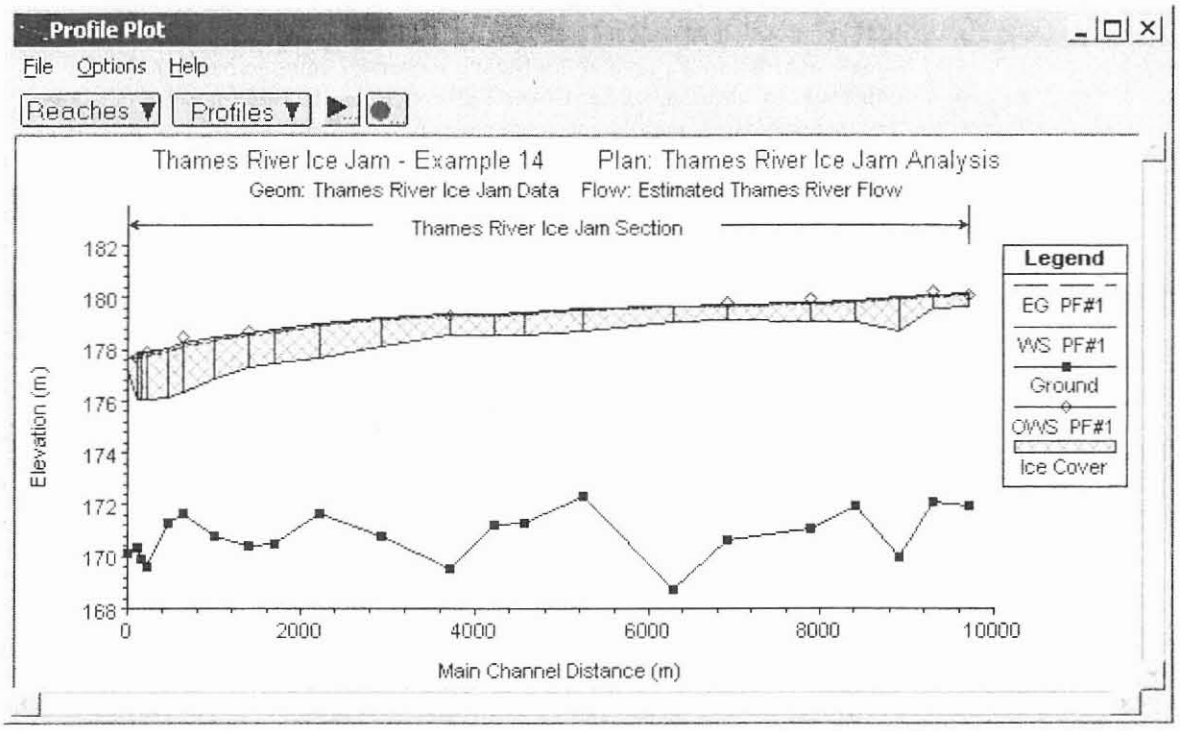


Figure 9.19 Profile plot with ice cover

Ice information can also be displayed in profile plots by selecting the **General Profile** option and then selecting **Variables** under the **Options** menu. This provides a number of ice variables, including ice volume in the channel, left, and right overbanks; ice thickness in the channel, left, and right overbanks; top of ice elevation in the channel, left, and right overbanks; and bottom of ice elevations in the channel, left, and right overbanks. These plots can all be viewed in different widow sizes and printed.

X-Y-Z Perspective Plot. As with the Cross Section plot, the ice cover is displayed by selecting **Variables** under the **Options** menu, then selecting the **Ice Cover** option. The ice thicknesses in the right overbank, main channel, and left overbank are displayed. The default color and fill pattern can be changed by the user by selecting **Lines and Symbols** under the Options menu.

Viewing Tabular Ice Information

Tabular information describing the results of the ice calculations can be displayed by selecting **Profile Summary Table** under the **View** menu on the HEC-RAS main window. Ice information is available directly by selecting the **Ice Cover** option under the **Std. Tables** menu of the Profile Table window. The Ice Cover option provides a table that includes the ice volume, ice thickness, and composite Manning's n value for the main channel, left overbank, and right overbank. In addition, the Ice Cover Table includes the water surface elevation and the cumulative ice volume starting from the downstream end of the channel. An example table of ice information is shown in Figure 9.20. Tables of ice information can also be created using the **Define Table** option under the **Options** menu of the Profile Table window.

Profile Output Table - Ice Cover

File Options Std. Tables User Tables Locations Help

HEC-RAS Plan: icejam River: Thames River Reach: Ice Jam Section Profile: PF#1 (Reload Data)

Reach	River Sta	W.S. Elev	Ice Thick LOB	Ice Thick Chan	Ice Thick ROB	Ice Vol Total	Ice Vol. LOB	Ice Vol. Chan	Ic
		(m)	(m)	(m)	(m)	(m3)	(m3)	(m3)	
Ice Jam Section	42000	180.12	0.00	0.50	0.00	496415.50		496415.50	
Ice Jam Section	41590	180.05	0.00	0.50	0.00	486141.30		486141.30	
Ice Jam Section	41190	179.94	0.00	1.37	0.00	468451.70		468451.70	
Ice Jam Section	40690	179.81	0.00	0.80	0.00	442029.90		442029.90	
Ice Jam Section	40180	179.75	0.00	0.75	0.00	418756.60		418756.60	
Ice Jam Section	39190	179.66	0.00	0.55	0.00	379948.00		379948.00	
Ice Jam Section	38560	179.63	0.00	0.60	0.00	362424.10		362424.10	
Ice Jam Section	37530	179.52	0.00	0.87	0.00	326998.50		326998.50	
Ice Jam Section	36670	179.38	0.00	0.93	0.00	297300.70		297300.70	
Ice Jam Section	36320	179.31	0.00	0.82	0.00	281821.20		281821.20	
Ice Jam Section	35820	179.26	0.00	0.76	0.00	260200.10		260200.10	
Ice Jam Section	35030	179.15	0.00	1.13	0.00	216909.70		216909.70	
Ice Jam Section	34320	178.90	0.00	1.36	0.00	171031.90		171031.90	
Ice Jam Section	33790	178.65	0.00	1.31	0.00	139049.20		139049.20	
Ice Jam Section	33490	178.54	0.00	1.35	0.00	120073.40		120073.40	

Figure 9.20 Ice Cover Table

Viewing Data Contained in an HEC-DSS File

The HEC-RAS software can write and read data to and from the HEC-DSS (Data Storage System) database. The steady flow portion of HEC-RAS can read flow data to be used as profile information, and can write water surface profiles, storage-outflow information, and rating curves. The unsteady flow portion of HEC-RAS can read complete hydrographs (stage and flow), as well as gate settings to be used during a simulation. Observed data contained in a DSS file can be attached to specific cross sections for comparison with computed results at those locations, and computed profiles and hydrographs are written to the DSS file during an unsteady flow simulation.

Because a DSS file can be used to share information between different HEC programs (such as HEC-HMS and HEC-RAS), it is often necessary to be able to view data contained within a DSS file. A DSS viewer is available from within the HEC-RAS software. To bring up the DSS viewer select **DSS Data** from the **View** menu of the main HEC-RAS window (Or press the button labeled **DSS** on the main window). When this option is selected a window will appear as shown in Figure 9.21.

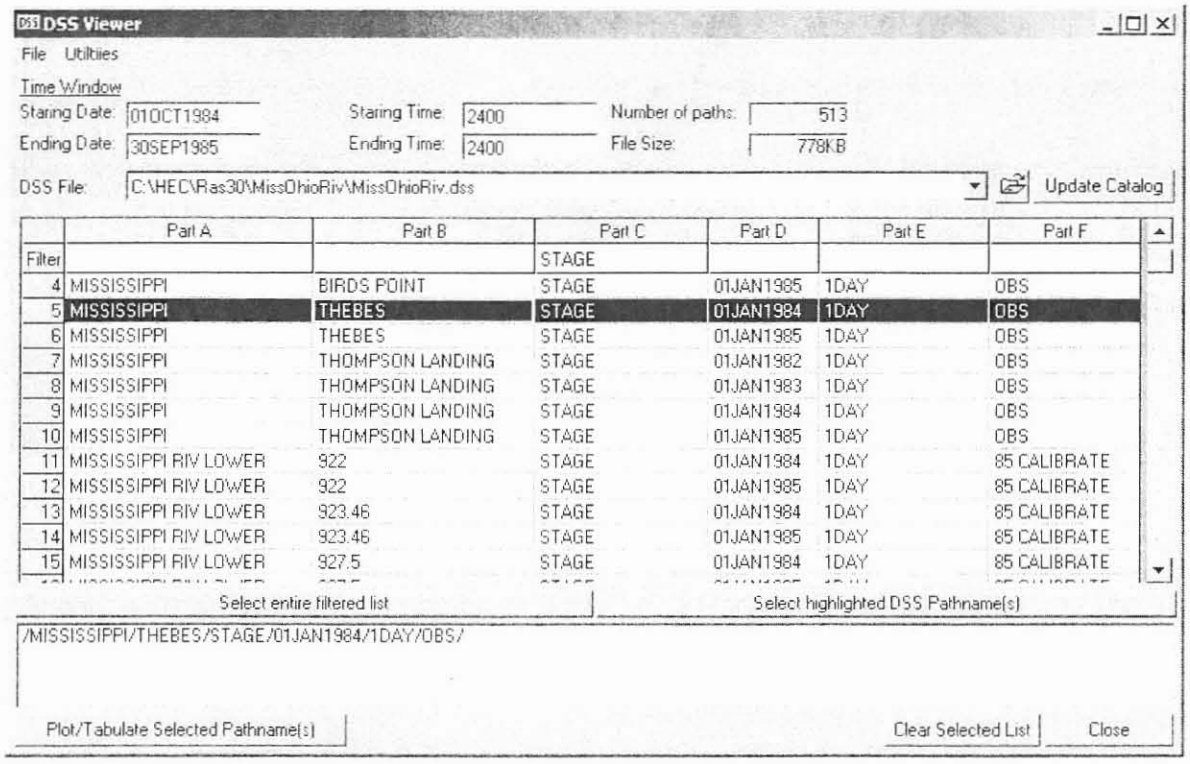


Figure 9.21 HEC-DSS Viewer Window

As shown in Figure 9.21, the user selects a DSS file by pressing the open file button located next to the DSS Filename field. When a DSS file is selected, a list of the available pathnames within that file will show up in the table. Each DSS pathname represents a record of data stored within the DSS file. The user can select one or more DSS pathnames to be plotted and/or tabulated. A pathname is selected by using the left mouse button to select a row(s) in the table, then the button labeled **Select highlighted DSS Pathnames** is pressed and the pathname shows up in the lower box. The final step is to hit the **Plot/Tabulate Selected Pathnames** button, and the data will be plotted. An example plot is shown in Figure 9.22.

As shown in Figure 9.22 there are two tabs on the window, one says **Plot** and the other says **Table**. By default the window comes up plotting the data. To view the data as a table, simply press the table tab.

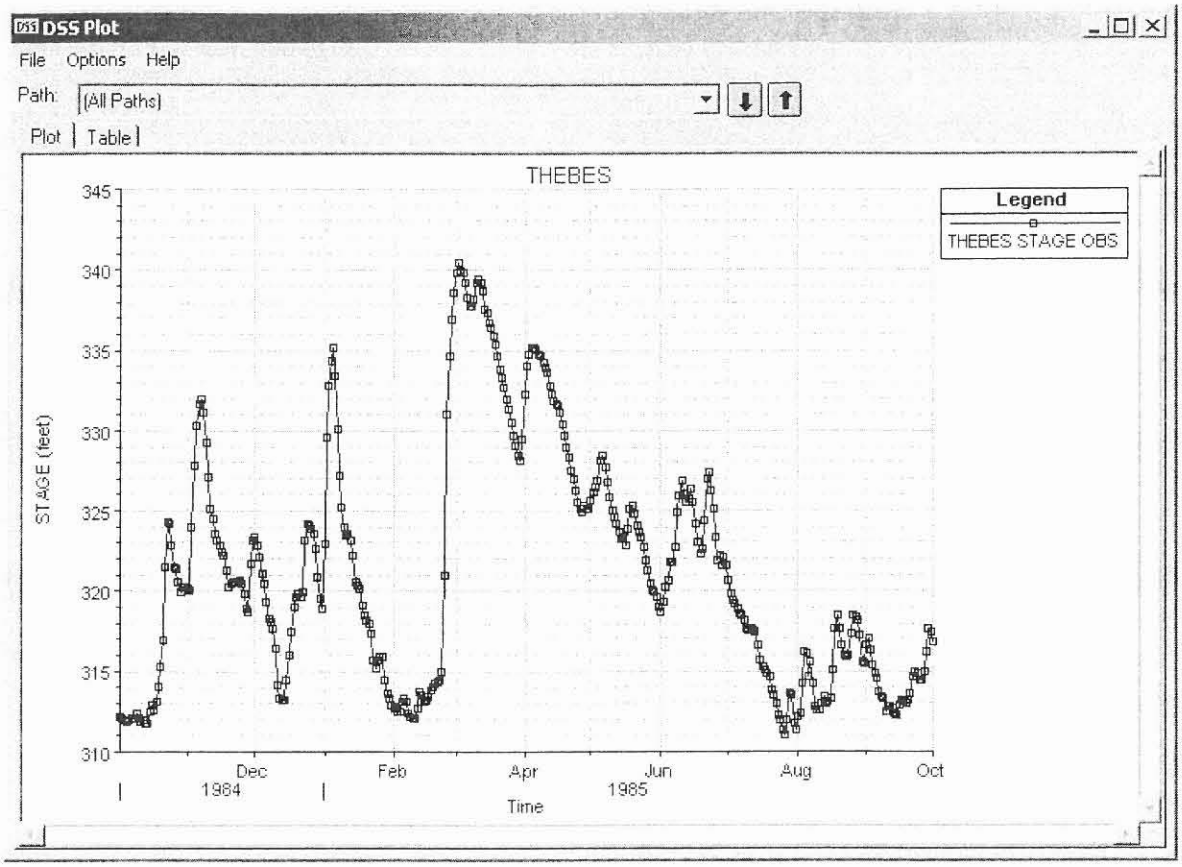


Figure 9.22 Example Plot From The HEC-RAS DSS Viewer

Data can be viewed from one or more DSS files simultaneously. The user simple opens one DSS file and picks the desired pathnames, then opens another DSS file and selects additional pathnames. When the Plot/Tabulate button is pressed, the data from both DSS files will be plotted and/or tabulated.

A few utilities are also available from the DSS viewer. These utilities include: Time Series Importer; Delete Selected Pathnames; and Squeeze the DSS file. The time series importer allows the user to enter regular interval time series data into a table, which can then be imported into a DSS file. To use this option select **Time Series Import** from the **Utilities** menu of the DSS Data Viewer. When this option is selected a window will appear as shown in Figure 9.23.

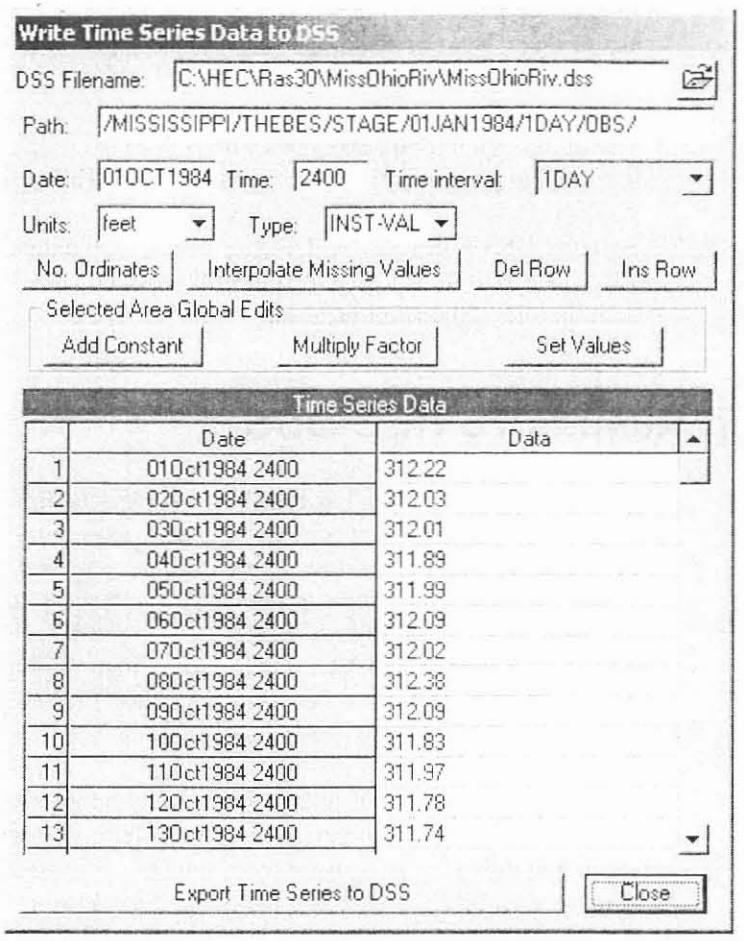


Figure 9.23 DSS Time Series Data Import Utility

As shown in Figure 9.23, the user first selects a DSS file to import data into. Next a DSS Pathname must be entered for the data to be written to the DSS file. The pathname parts are separated with a "/" between each pathname part. Some parts can be left blank, but the B and C part must be entered at a minimum. Next the user enters the date and time of the first data point, as well as the interval of the data (the interval is selected from the available DSS intervals). Next the data units and data type are selected from the drop down lists. If the lists do not contain the units of your data you can enter them directly into the field. The data is then entered into the table at the bottom. You can cut and paste information into this table, using the standard windows keys of Ctrl-C for cut, and Ctrl-V for paste. There are buttons available to perform the following tasks: set the number of rows in the table (the default is 99); linearly interpolate missing values; delete a row; insert a row; add a constant to a highlighted section of the table; multiply the highlighted section by a factor; and set a highlighted section to a specific value.

The utility labeled **Delete Selected Paths** is used to delete data from the DSS file. The user simply selects the pathnames they want to delete, then selects this option from the **Utilities** menu. A window will appear to asking if you

are sure you want to delete the selected pathnames. If you answer **OK**, then the data will be deleted from the DSS file.

The utility labeled **Squeeze DSS File** is used to compress the DSS file, such that it takes significantly less hard disk space. This is a convenient function if you are working with very large DSS files. To use this option just select **Squeeze DSS File** from the **Utilities** menu. A window will come up asking you if you want to squeeze the currently opened DSS file. If you answer **OK** then the file will be compressed.

Exporting Results To HEC-DSS

The HEC-RAS software has the ability to export a limited set of results to a HEC-DSS file for both steady and unsteady flow simulations. When performing an Unsteady flow simulation, the program automatically writes stage and flow hydrographs to the DSS file, but only for the user-selected hydrograph output locations. Water surface profiles are also automatically written to the DSS file. The profiles are written for the user selected detailed output interval, as well as the overall maximum water surface profile (profile of the maximum stage at every cross section).

Once a steady flow or unsteady flow simulation is performed, the user can write the following information to a DSS file: water surface profiles; computed rating curves; and storage-outflow information. To export computed results to a DSS file the user selects **Export To HEC-DSS** from the **File** menu of the main HEC-RAS window. When this option is selected a window will appear as shown in Figure 9.24.

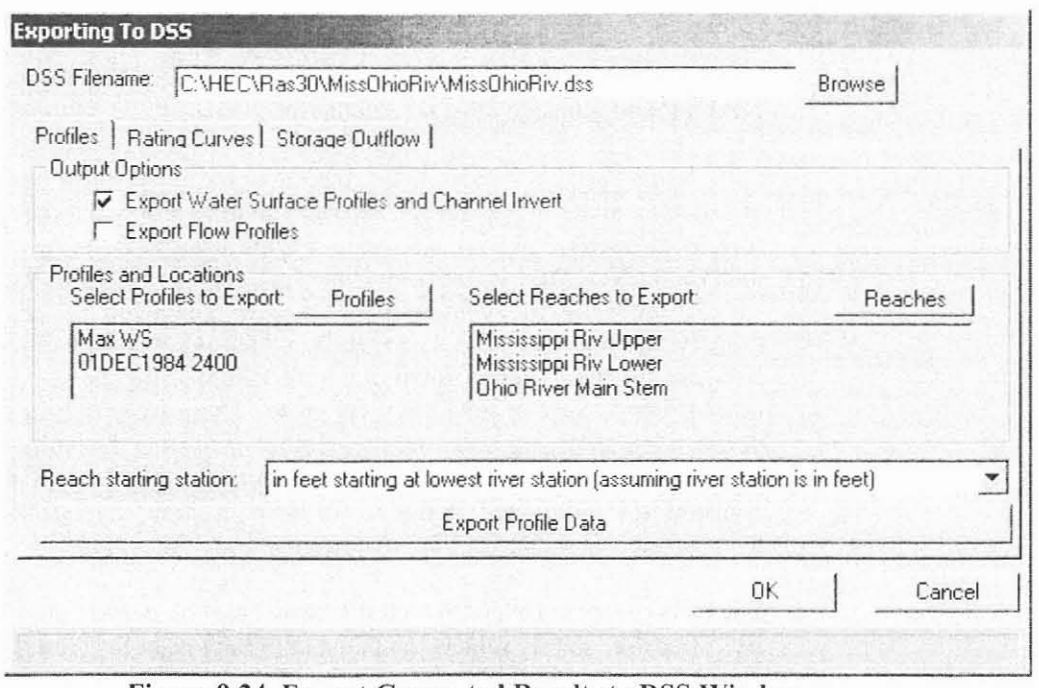


Figure 9.24 Export Computed Results to DSS Window

As shown in Figure 9.24, there are three tabs on the window; one for profiles, rating curves, and storage outflow. To export computed water surface profiles, select the **Profiles** tab from the window. Select the type of profiles that you want to export (water surface elevations or flow). Next select the specific profiles to be exported, as well as the reaches that you want to have profiles for. Select how you want the stationing to be labeled. This is accomplished by selecting one of the options under the field labeled **Reach Starting Station**. The user can have the river stationing labeled in feet or miles, and have it start at zero or whatever the magnitude is of the most downstream cross section. The final option is to press the **Export Profile Data** button, and the data will then be written to the DSS file.

To write computed rating curves to the DSS file select the **Rating Curve** tab. When the rating curve tab is selected, the window will change to what is shown in Figure 9.25.

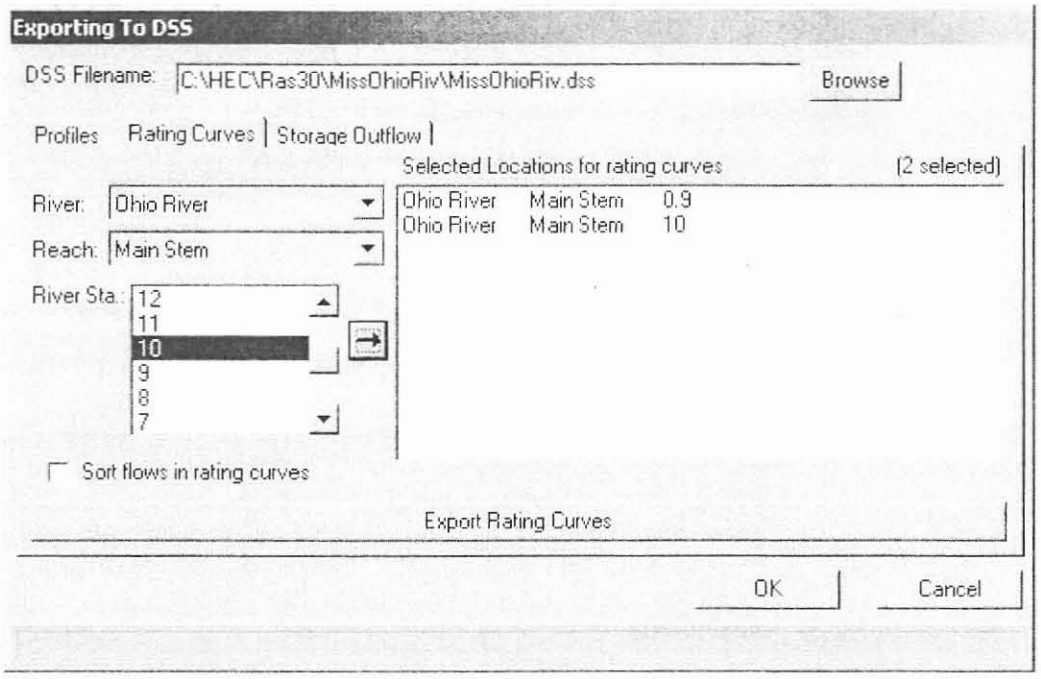


Figure 9.25 Exporting Computed Rating Curves to HEC-DSS

As shown in Figure 9.25, to export a computed rating curve to DSS, select the river, reach, and river stations that you want to have exported to the DSS file. Then simply press the **Export Rating Curves** button to have the program write the data to the DSS file. If your profiles are not in the order from lowest flow to highest flow, turn on the option that says **Sort flows in rating curve**. This option will ensure that the curve is written in the order of increasing flow rate.

The HEC-RAS program computes cumulative storage volumes for each of the water surface profiles. This information can be used for hydrologic

routing in a hydrology model such as HEC-HMS or HEC-1. The HEC-RAS program allows the user to write out storage versus volume information to a DSS file. To use this option select the **Storage Outflow** tab from the Export to DSS window. When this option is selected a window will appear as shown in Figure 9.26.

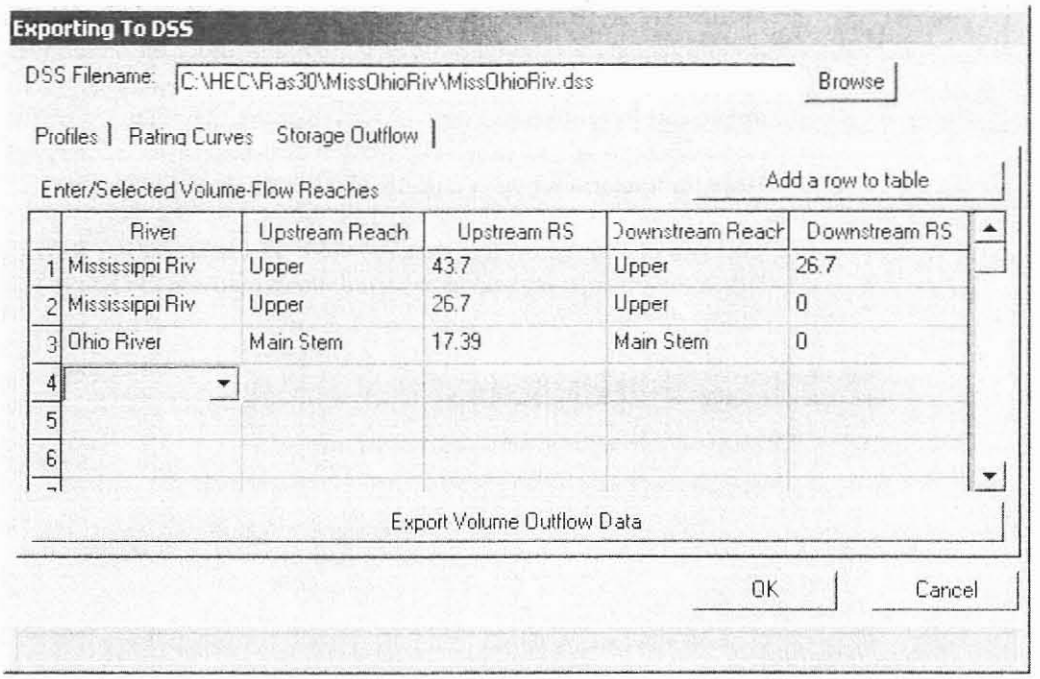


Figure 9.26 Exporting Storage-Outflow Information to HEC-DSS

As shown in Figure 9.26, the user selects the River, upstream reach, upstream river station, downstream reach, and downstream river station to completely define a routing reach in which they want to have storage-outflow information written to the DSS file. This can be done for as many reaches as you want within the model. After all of the reaches are defined, simply press the button labeled **Export Volume Outflow Data** to write the information to the DSS file.

CHAPTER 10

Performing a Floodplain Encroachment Analysis

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

FEMA 37 defines a floodway "...as the channel of a river or other watercourse and the adjacent land areas that must be reserved in order to discharge the base flood without cumulatively increasing the water-surface elevation by more than a designated height." Normally, the base flood is the one-percent chance event (100-year recurrence interval), and the designated height is one foot, unless the state has established a more stringent regulation for maximum rise. The floodway is usually determined by an encroachment analysis, using an equal loss of conveyance on opposite sides of the stream. For purposes of floodway analysis, the floodplain fringe removed by the encroachments is assumed to be completely blocked.

HEC-RAS contains five optional methods for specifying floodplain encroachments. For information on the computational details of each of the five encroachment methods, as well as special considerations for encroachments at bridges, culverts, and multiple openings, see Chapter 10 of the HEC-RAS hydraulics reference manual. This chapter describes how to enter floodplain encroachment data, how to perform the encroachment calculations, viewing the floodplain encroachment results, and how to perform a floodplain encroachment analysis within the unsteady flow computations module.

Contents

- General
- Entering Floodplain Encroachment Data
- Performing the Floodplain Encroachment Analysis
- Viewing the Floodplain Encroachment Results
- Floodplain Encroachments With Unsteady Flow

General

The HEC-RAS floodplain encroachment procedure is based on calculating a natural profile (existing conditions geometry) as the first profile in a multiple profile run. Other profiles, in a run, are calculated using various encroachment options, as desired. Before performing an encroachment analysis, the user should have developed a model of the existing river system. This model should be calibrated to the fullest extent that is possible. Verification that the model is adequately modeling the river system is an extremely important step before attempting to perform an encroachment analysis.

Currently, the HEC-RAS steady flow program has 5 methods to determine floodplain encroachments. These methods are:

Method 1 -	User enters right and left encroachment stations
Method 2 -	User enters fixed top width
Method 3 -	User specifies the percent reduction in conveyance
Method 4 -	User specifies a target water surface increase
Method 5 -	User specifies a target water surface increase and maximum change in energy

For unsteady flow analysis, only method one has been implemented so far in HEC-RAS. For a detailed discussion on each of these methods, the user is referred to Chapter 10 of the **Hydraulic Reference Manual**.

The goal of performing a floodplain encroachment analysis is to determine the limits of encroachment that will cause a specified change in water surface elevation. To determine the change in water surface elevation, the program must first determine a natural profile with no encroachments. This base profile is typically computed using the one percent chance discharge. The computed profile will define the floodplain, as shown in Figure 10.1. Then, by using one of the 5 encroachment methods, the floodplain will be divided into two zones: the floodway fringe and the floodway. The floodway fringe is the area blocked by the encroachment. The floodway is the remaining portion of the floodplain in which the one-percent chance event must flow without raising the water surface more than the target amount.

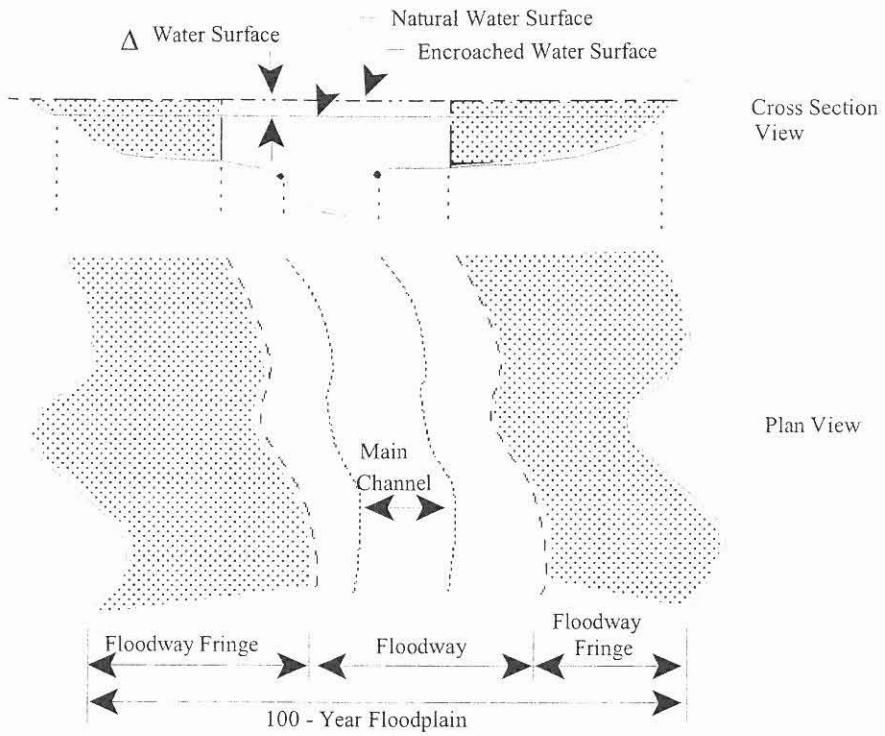


Figure 10.1 Floodway Definition Sketch

Entering Floodplain Encroachment Data

Within HEC-RAS, the data for performing a steady flow floodplain encroachment analysis are entered from the Steady Flow Analysis window. Encroachment information is not considered as permanent geometry or flow data, and is therefore not entered as such. The encroachment information is saved as part of the existing Plan data.

To bring up the floodplain encroachment data window, select the **Encroachments** option from the **Options** menu of the Steady Flow Analysis window. When this option is selected an Encroachment window will appear as shown in Figure 10.2 (except yours will be blank when you first open it).

As shown in Figure 10.2, there are several pieces of data that the user must supply for an encroachment analysis. The encroachment analysis can only be performed for profiles 2 through 15 (or what ever number has been set by the user in the flow data editor). Encroachments are not performed on profile one because most of the encroachment methods rely on having a base profile for comparison.

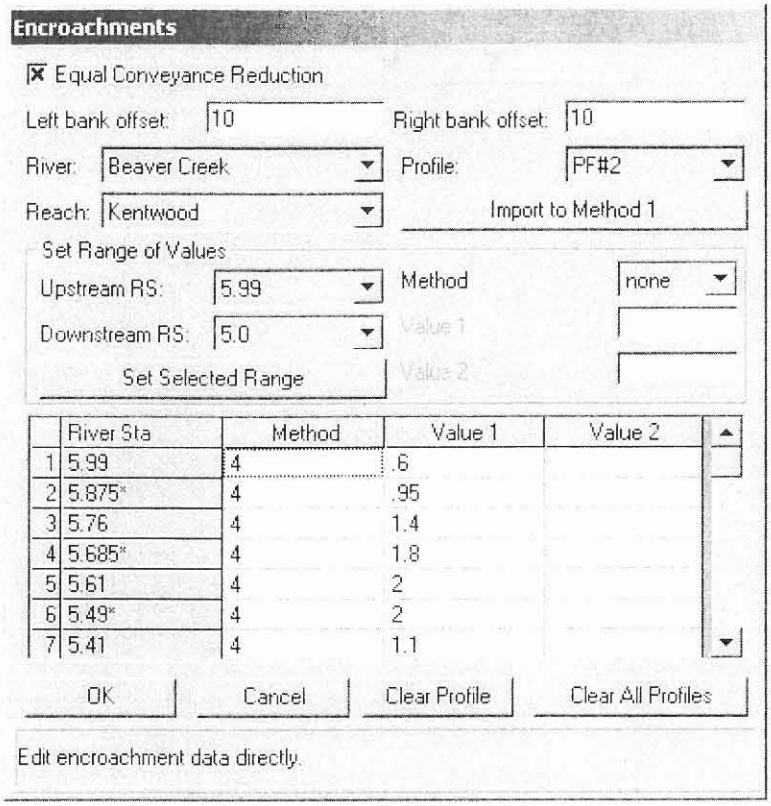


Figure 10.2 Floodplain Encroachment Data Editor

The data for an encroachment analysis should be entered in the following manner:

Global Information. Global information is data that will be applied at every cross section for every profile computed. The first piece of global information is the **Equal Conveyance Reduction** selection box at the top of the Encroachment data editor window. Equal conveyance reduction applies to encroachment methods 3, 4, and 5. When this is turned on, the program will attempt to encroach, such that an equal loss of conveyance is provided on both sides of the stream. If this option is turned off, the program will encroach by trying to maintain a loss in conveyance in proportion to the distribution of natural overbank conveyance. The default is to have equal conveyance reduction turned on.

The second item under global information is the **Left bank offset** and the **Right bank offset**. The left and right offsets are used to establish a buffer zone around the main channel for further limiting the amount of the encroachments. For example, if a user established a right offset of 5 feet and a left offset of 10 feet, the model will limit all encroachments to 5 feet from the right bank station and 10 feet from the left bank station. The default is to have no right or left offset, this will allow the encroachments to go up to the main channel bank stations, if necessary.

River, Reach and River Station Selection Boxes. The next piece of data for the user to select is the river and reach in which to enter encroachment data. The user is limited to seeing one reach at a time on the encroachment data editor. Once a reach is selected, the user can then enter a **Starting and Ending River Station** to work on. By default, the program selects all the sections in the reach. The user can change this to any range of cross sections within the reach.

Profile. Next, the user should select a profile number to work on. Profiles are limited to 2 through the maximum number set in the currently opened flow data (e.g., 2 through 4, if the user has set 4 profiles in the flow data editor). The user can not set encroachments for profile 1.

Method and Target Values. The next step is to enter the desired encroachment method to be used for the currently selected profile. Once a method is selected, the data entry boxes that corresponds to that method will show up below the method selection box. Some of the methods require only one piece of data, while others require two. The user should then enter the required information that corresponds to the method that they have selected. For example, if the user selects encroachment method 4, only one piece of information is required, the target change in water surface elevation. The available encroachment methods in HEC-RAS are:

Method 1 -	User enters right and left encroachment station
Method 2 -	User enters a fixed top width
Method 3 -	User specifies the percent reduction in conveyance
Method 4 -	User specifies a target water surface increase
Method 5 -	User specifies target water surface increase and maximum change in energy

Set Selected Range. Once the encroachment method is selected, and its corresponding data are entered, the user should press the **Set Selected Range** button. Pressing this button will fill in the table below with the selected range of river stations; the selected method; and the corresponding data for the method. Note that, if the selected method only has one data item, that method's data will go under the **Value 1** column of the table. If the selected method has two data items, the first goes into the **Value 1** column and the second goes into the **Value 2** column. Once the data is put into the table, the user can change the method and corresponding data values directly from the table.

At this point the user should repeat these tasks until all of the encroachment data are entered (i.e., for all the reaches and locations in the model, as well as all of the profiles for which the user wants to perform the encroachment analysis). Once all of the encroachment data are entered, the user presses the **OK** button and the data will be applied and the window will close. The user can return to the encroachment window and edit the data at any time. The encroachment data are not saved to the hard disk at this time, they are only saved in memory. To save the data to the hard disk, the user should either select **Save Project** from the File menu of the main HEC-RAS window, or

select **Save Plan** from the File menu of the Steady Flow Analysis window.

The **Import Method 1** option, allows the user to transfer the computed encroachment stations from a previous run (output file) to the input data for a future run. For example, if the user performs a preliminary encroachment analysis using any of the methods 2 through 5, they may want to convert the results from one of the runs to a method 1 encroachment method. This will allow the user to further define the floodway, using method 1, without having to enter all of the encroachment stations. The import of encroachment stations, in this manner, is limited to the results of a single encroachment profile for each reach.

Performing The Floodplain Encroachment Analysis

The HEC-RAS floodway procedure is based on calculating a natural profile (no encroachments) as the first profile of a multiple profile run. Subsequent profiles are calculated with the various encroachment options available in the program.

In general, when performing a floodway analysis, encroachment methods 4 and 5 are normally used to get a first cut at the encroachment stations. Recognizing that the initial floodway computations may provide changes in water surface elevations greater, or less, than the "target" increase, initial computer runs are usually made with several "target" values. The initial computer results should then be analyzed for increases in water surface elevations, changes in velocities, changes in top width, and other parameters. Also, plotting the results with the X-Y-Z perspective plot, or onto a topo map, is recommended. From these initial results, new estimates can be made and tested.

After a few initial runs, the encroachment stations should become more defined. Because portions of several computed profiles may be used, the final computer runs are usually made with encroachment Method 1 defining the specific encroachment stations at each cross section. Additional runs are often made with Method 1, allowing the user to adjust encroachment stations at specific cross sections to further define the floodway.

While the floodway analysis generally focuses on the change in water surface elevation, it is important to remember that the floodway must be consistent with local development plans and provide reasonable hydraulic transitions through the study reach. Sometimes the computed floodway solution, that provides computed water surfaces at or near the target maximum, may be unreasonable when transferred to the map of the actual study reach. If this occurs, the user may need to change some of the encroachment stations, based on the visual inspection of the topo map. The floodway computations should be re-run with the new encroachment stations to ensure that the target maximum is not exceeded.

Viewing the Floodplain Encroachment Results

Floodplain encroachment results can be viewed in both graphical and tabular modes. Graphically, the encroachment results show up on the cross section plots as well as the X-Y-Z Perspective plot. An example cross-section plot is shown in Figure 10.3.

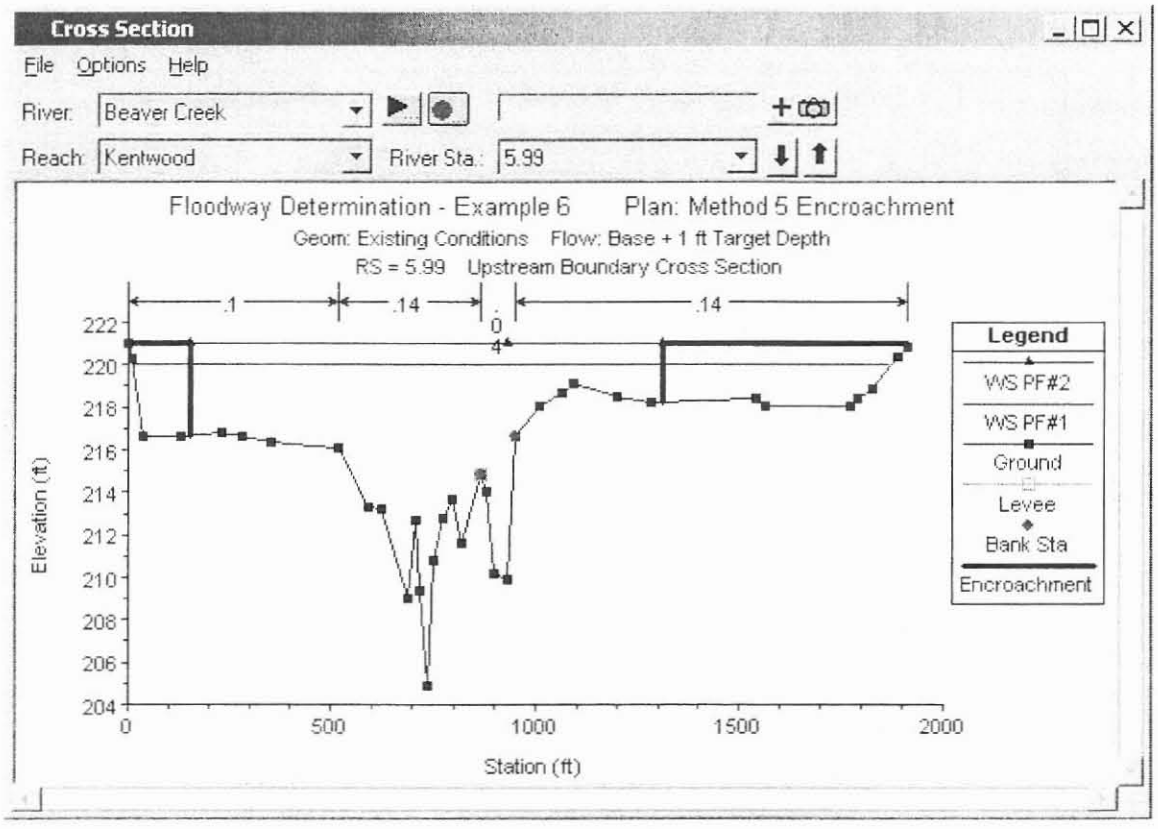


Figure 10.3 Example Cross Section Plot With Encroachments

As shown in Figure 10.3, the encroachments are plotted as outlined blocks. In this example, the water surface profile for the base run (first profile) is plotted along with one of the encroached profiles. The user can plot as many profiles as they wish, but it may become a little confusing with several sets of encroachments plotted at they same time.

Another type of graphic that can be used to view the encroachments is the X-Y-Z perspective plot, an example is shown in Figure 10.4. In this example, the base profile (profile 1) as well as one of the encroached profiles is plotted at the same time over a range of cross sections. This type of plot allows the user to get a reach view of the floodplain encroachment. The user can quickly see if the encroachments transition smoothly or if they are erratic. In general, the final encroachments should have a consistent and smooth

transition from one cross section to the next. With the assistance of this type of plot, the user may want to further refine the final encroachment stations and re-run the model.

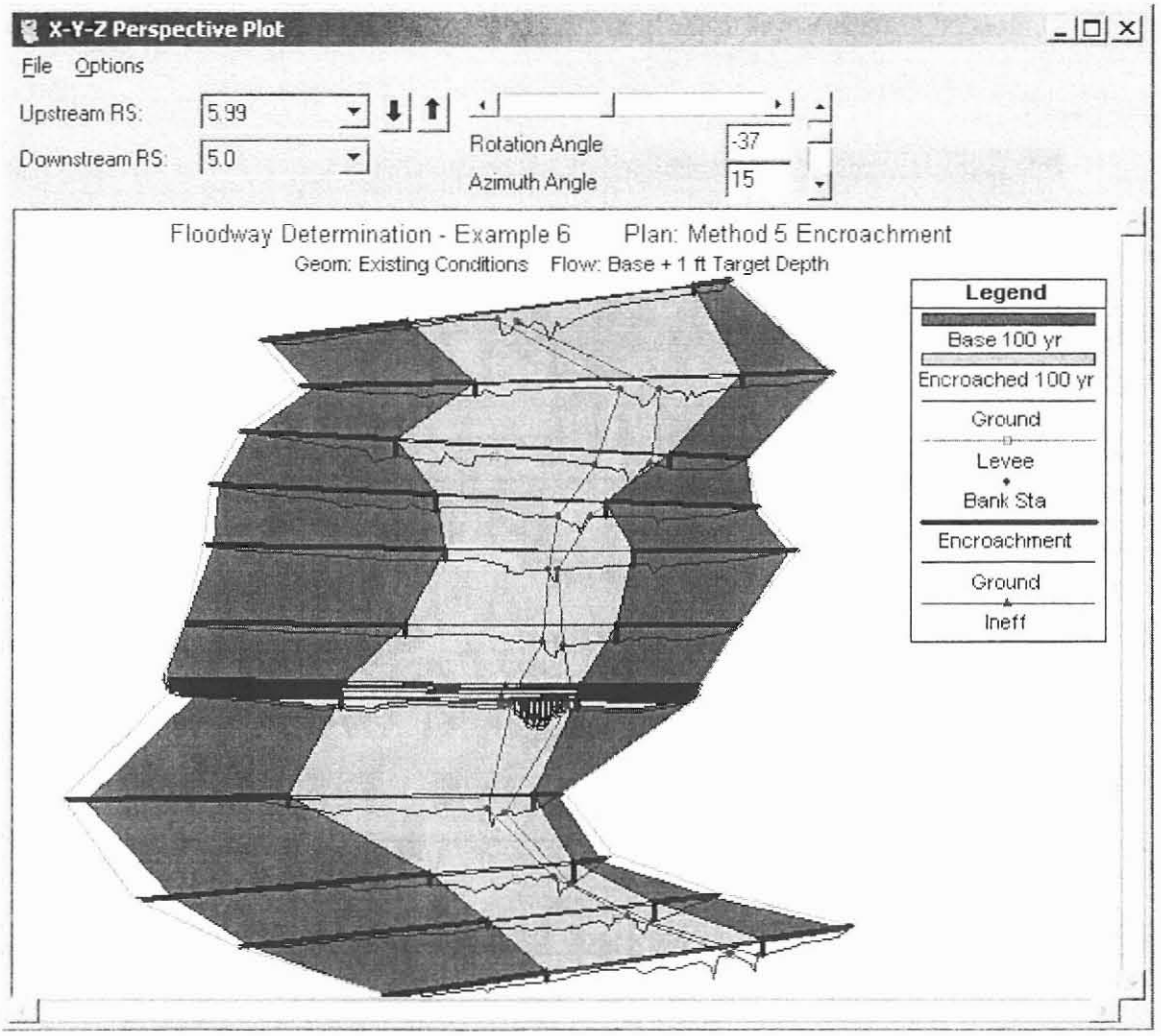


Figure 10.4 Example X-Y-Z Perspective Plot with Base and Encroached Profiles

Encroachment results can also be viewed in a tabular mode from the Profile Output Tables. Select **Profile Table** from the **View** menu of the main HEC-RAS window. When the table comes up, the user can select from three different pre-defined encroachment tables. To bring up one of the encroachment tables, select **Encroachment 1** from the **Std. Tables** menu on the Profile table window. An example of Encroachment 1 table is shown in Figure 10.5. The table shows the basic encroachment results of: computed water surface elevation; change in water surface from the base profile; the computed energy; top width of the active flow area; the flow in the left overbank, main channel, and right overbank; the left encroachment station; the station of the left bank of the main channel; the station of the right bank of the main channel; and the right encroachment station.

Profile Output Table - Encroachment 1

File Options Std. Tables Help

HEC-RAS Plan: M5 River: Beaver Creek Reach: Kentwood — Reload Data

Reach	River Sta	W.S. Elev (ft)	Prof Delta WS (ft)	E.G. Elev (ft)	Top Wdth Act (ft)	Q Left (cfs)	Q Channel (cfs)	Q Right (cfs)	Enc Sta L (ft)	Ch Sta L (ft)	Ch Sta R (ft)
Kentwood	5.4 BR U	217.44		217.68	1847.04	849.46	10817.92	2376.36		450.00	647.00
Kentwood	5.4 BR U	217.39	-0.04	217.79	807.60	65.95	12136.13	1797.88	420.85	450.00	647.00
Kentwood	5.4 BR D	217.44		217.68	1824.00	858.86	10867.73	2317.15		450.00	647.00
Kentwood	5.4 BR D	217.39	-0.04	217.79	807.60	67.29	12148.29	1784.38	420.85	450.00	647.00
Kentwood	5.39	215.61		216.04	1702.76	1073.43	10253.83	2672.74		450.00	647.00
Kentwood	5.39	216.35	0.73	216.86	807.60	113.48	11602.49	2284.04	420.85	450.00	647.00
Kentwood	5.24*	214.63		214.77	1633.15	2336.07	2507.27	9156.67		200.30	257.00
Kentwood	5.24*	215.63	0.99	215.75	845.19	1922.36	2527.11	9550.53	101.99	200.30	257.00
Kentwood	5.13	213.33		213.76	1425.37	1102.78	4954.85	7942.37		155.00	213.00
Kentwood	5.13	214.34	1.01	214.92	535.14	212.93	5788.66	7998.41	145.00	155.00	213.00
Kentwood	5.065*	212.55		212.89	1782.32	1624.48	5352.58	7022.95		274.50	365.50
Kentwood	5.065*	213.54	0.99	214.03	642.15	167.70	6493.35	7338.95	264.50	274.50	365.50
Kentwood	5.0	211.80		212.05	1925.36	2217.73	5187.01	6595.27		394.00	518.00
Kentwood	5.0	212.80	1.00	213.18	912.57	127.69	6544.90	7327.41	384.00	394.00	518.00

Difference in WS between current profile and WS for first profile.

Figure 10.5 Example of the Encroachment 1 Standard Table

Encroachment 2 table provides some additional information that is often used when plotting the encroachments onto a map. This table includes: the change in water surface elevations from the first profile; the top width of the active flow area; the percentage of conveyance reduction in the left overbank; the left encroachment station; the distance from the center of the main channel to the left encroachment station; the station of the center of the main channel; the distance from the center of the main channel to the right encroachment station; the right encroachment station; and the percentage of conveyance reduction in the right overbank. An example of the Encroachment 2 standard table is shown in Figure 10.6.

Profile Output Table - Encroachment 2

File Options Std. Tables Help

HEC-RAS Plan: Method 1 River: Beaver Creek Reach: Kentwood

Reload Data

Reach	River Sta	Prof Delta WS	Top Wdth Act	K Perc L	Enc Sta L	Dist Center L	Center Station	Dist Center R	Enc Sta R	K Perc R	Encr WD
		(ft)	(ft)		(ft)	(ft)	(ft)	(ft)	(ft)		(ft)
Kentwood	5.99		1862.66				907.00				
Kentwood	5.99	1.00	885.42	8.96	233.00	674.00	907.00	211.42	1118.42	10.01	885.42
Kentwood	5.875*		1797.64				678.25				
Kentwood	5.875*	0.98	928.58	12.53	327.07	351.18	678.25	577.40	1255.65	12.56	928.58
Kentwood	5.76		1765.50				449.50				
Kentwood	5.76	0.86	847.98	13.62	341.00	108.50	449.50	739.48	1188.98	14.84	847.98
Kentwood	5.685*		1873.52				647.25				
Kentwood	5.685*	0.72	789.38	17.40	464.82	182.43	647.25	606.95	1254.20	16.00	789.38
Kentwood	5.61		1989.05				845.00				
Kentwood	5.61	0.62	707.62	20.61	609.60	235.40	845.00	472.22	1317.22	16.20	707.62
Kentwood	5.49*		1910.09				647.00				
Kentwood	5.49*	0.43	752.77	20.64	477.33	169.67	647.00	583.10	1230.10	15.40	752.77
Kentwood	5.41		1847.04				548.50				
Kentwood	5.41	0.11	617.70		440.00	108.50	548.50	509.20	1057.70		617.70
Kentwood	5.4 BR U		1847.04				548.50				
Kentwood	5.4 BR U	0.11	617.70		440.00	108.50	548.50	509.20	1057.70		617.70

Figure 10.6 Example of the Encroachment 2 Standard Table

The last encroachment table, Encroachment 3, provides the minimum floodway data for reporting. This table includes: the active flow top width; the flow area (including any ineffective flow area); the average velocity of the entire cross section; the computed water surface elevation; the base water surface elevation (profile 1); and the change in water surface from the first profile. An example of this table is shown in Figure 10.7

Profile Output Table - Encroachment 3

File Options Std. Tables Help

HEC-RAS Plan: Method 1 River: Beaver Creek Reach: Kentwood Reload Data

Reach	River Sta	Top Wdth Act	Area	Vel Total	W.S. Elev	Base WS	Prof Delta WS
		(ft)	(sq ft)	(ft/s)	(ft)	(ft)	(ft)
Kentwood	5.99	1862.66	6609.02	2.12	220.00	220.00	
Kentwood	5.99	885.42	5564.08	2.52	221.00	220.00	1.00
Kentwood	5.875*	1797.64	7068.82	1.98	218.99	218.99	
Kentwood	5.875*	928.58	5276.75	2.65	219.97	218.99	0.98
Kentwood	5.76	1765.50	9092.73	1.54	218.46	218.46	
Kentwood	5.76	847.98	5913.15	2.37	219.32	218.46	0.86
Kentwood	5.685*	1873.52	9248.86	1.51	218.23	218.23	
Kentwood	5.685*	789.38	5530.05	2.53	218.95	218.23	0.72
Kentwood	5.61	1989.05	9499.42	1.47	218.09	218.09	
Kentwood	5.61	707.62	5352.12	2.62	218.71	218.09	0.62
Kentwood	5.49*	1910.09	9447.83	1.48	217.91	217.91	
Kentwood	5.49*	752.77	5179.35	2.70	218.34	217.91	0.43
Kentwood	5.41	1847.04	8985.57	1.56	217.44	217.44	
Kentwood	5.41	617.70	4043.15	3.46	217.54	217.44	0.11
Kentwood	5.4 BR U	1847.04	2533.60	5.53	217.44	217.44	
Kentwood	5.4 BR U	617.70	1979.08	7.07	217.54	217.44	0.11
Kentwood	5.4 BR D	1824.00	2522.19	5.55	217.44	217.44	
Kentwood	5.4 BR D	617.70	1979.08	7.07	217.54	217.44	0.11

Top width of the wetted cross section, not including ineffective flow.

Figure 10.7 Example of the Encroachment 3 Standard Table

Floodway Encroachments With Unsteady Flow

Encroachment analyses can also be performed with the unsteady flow computations module within HEC-RAS. However, only method one (user placed encroachments) has been added to the unsteady flow computations. A suggested methodology for performing an encroachment analysis with an unsteady flow model is the following:

1. First, develop the unsteady flow model of the river system and calibrate it to the extent possible.

2. Develop an unsteady flow plan of the 100 yr event in order to establish the base floodplain.

3. Develop a steady flow plan that incorporates the peak flows from the unsteady flow run as the 100 yr event for the model. Set up the model for two profiles with the same flows.

4. Perform a steady flow encroachment analysis using the available steady flow encroachment methods to calculate an approximate floodway.

5. Copy the unsteady flow plan to a new plan (using the Save As option), and give it a name that represents the encroached plan.

6. Adjust the boundary conditions file to reflect an increased water surface elevation at the downstream boundary for the range of possible flows. If using a rating curve, you will need to develop a new rating to reflect the encroached condition at the downstream boundary. If you are using normal depth or critical depth, no change is necessary, since the program will calculate a new water surface with the encroachments.

7. Go to the **Options** menu of the unsteady flow analysis window and select **Unsteady Encroachments**. This will bring up the Unsteady flow Encroachment editor shown in Figure 10.8.

8. Import the final encroachments from the steady flow encroachment run in to the unsteady flow encroachment editor. This is accomplished by pressing the button labeled "**Get Encroachments From Steady Flow Plan**", and then selecting the appropriate plan and profile number from the steady flow encroachment analysis.

9. Run the unsteady flow model with the encroachments and compare the output of the encroached unsteady flow plan with the output from the base unsteady flow plan.

10. Adjust the encroachments as necessary to stay within the limits for increased water surface elevations. Re-run the unsteady flow model. Repeat this process until a final floodway is achieved.

Unsteady Encroachments

River: Beaver Creek

Reach: Kentwood

Get Encroachments from Steady Flow Plan ...

Selected Area Global Edits

Add Constant | Multiply Factor | Set Values

Unsteady Encroachments (Method 1 Style Encroachments)

	RS	Left Station	Right Station
1	5.99	330.12	958
2	5.95166*	334.79	1095.31
3	5.91333*	337.55	1207.04
4	5.875*	332.43	1251.37
5	5.83666*	319.27	1261.49
6	5.79833*	300.81	1262.91
7	5.76	283.43	1258.91
8	5.7225*	313.98	1298.11
9	5.685*	358.24	1341.27
10	5.6475*	414.51	1380.63
11	5.61	481.56	1413.29
12	5.57666*	487.35	1390.32
13	5.54333*	454.73	1350.93
14	5.51*	436.56	1308.04
15	5.47666*	432.72	1261.06

OK | Cancel

Figure 10.8 Unsteady Flow Encroachment Editor.

CHAPTER 11

Troubleshooting With HEC-RAS

For a steady flow analysis, the HEC-RAS software is designed to continue its computations all the way through completion, even when the user has entered poor data. Because of this, the fact that the program executes a complete run does not necessarily mean that the results are good. The user must carefully review the results to ensure that they adequately represent the study reach and that they are reasonable and consistent. The HEC-RAS software is an engineering tool, it is by no means a replacement for sound engineering.

The HEC-RAS software contains several features to assist the user in the development of a model; debugging problems; and the review of results. These features include: built in data checking; an Errors, Warnings, and Notes system; and a computational Log Output file. In addition to these features, the user can use the graphical and tabular output to review the results and check the data for reasonableness and consistency. Most of the information contained within this chapter is based on performing a steady flow analysis. Much of the information is also useful when performing an unsteady flow analysis. More assistance for solving unsteady flow stability problems can be found in Chapter 8 of this manual.

Built in Data Checking

The HEC-RAS user interface has two types of built in data checking. The first type of data checking is performed as the user enters the data. Each data field of the data entry editors has some form of data checking. The second type of data checking occurs when the user starts the steady flow or unsteady flow computations. When the user presses the compute button, on the steady flow or unsteady flow analysis window, the entire data set is processed through several data checks before the computations begin. A detailed discussion of each of these two data checking features is described below.

Checking the Data as it is Entered

This type of data checking occurs whenever the user enters data into a single data field or table. Once the user leaves a particular data entry field or table, the program will automatically check that data for reasonableness. The following is a list of some of the types of data checks that are performed:

1. Minimum and maximum range checking for variables.

2. Alpha and numeric data checks. This is done to ensure that the right type of data is entered in each field.

3. Increasing order of station for cross sections, bridge deck/roadway, and abutments.

4. Data consistency checks (i.e., when the main channel bank stations are entered, the program checks to see if they exist in the cross section station and elevation data).

5. Data deletion warnings. When you delete data the software will give you a warning before it is deleted.

6. File management warnings (i.e., program will give you a chance to save the data to the hard disk before the program is closed, or a different data set is opened).

7. Data geometry checks (i.e., when a bridge deck/roadway is entered, the program checks to ensure that the deck/roadway intersects with the ground data).

Data Checking Before Computations are Performed

The second type of data checking is performed to evaluate the completeness and consistency of the data. This type of data checking occurs before the computations take place. When the user presses the **Compute** button on the Steady Flow or Unsteady Flow Analysis window, the program will perform a series of data checks before the computations are allowed to proceed. If any data errors are found, the program will not perform the computations. The following is a list of some of the types of checks that are made during this time:

1. Data completeness. These data checks insure that all of the required data exists for the entire data set. If any missing data are found, a complete list of all the missing data and their specific locations is displayed on the screen. An example of this is shown in Figure 11.1.

2. Data consistency. This type of data checking is performed to ensure that the data is consistent with the computations that are being requested. For example, if the user asks to perform a mixed flow regime computation, the program checks to ensure that upstream as well as downstream boundary conditions have been specified. Likewise, if an encroachment analysis is requested, the program checks to ensure that the number of profiles lines up with the number specified in the encroachment data. There are several other checks of this type.

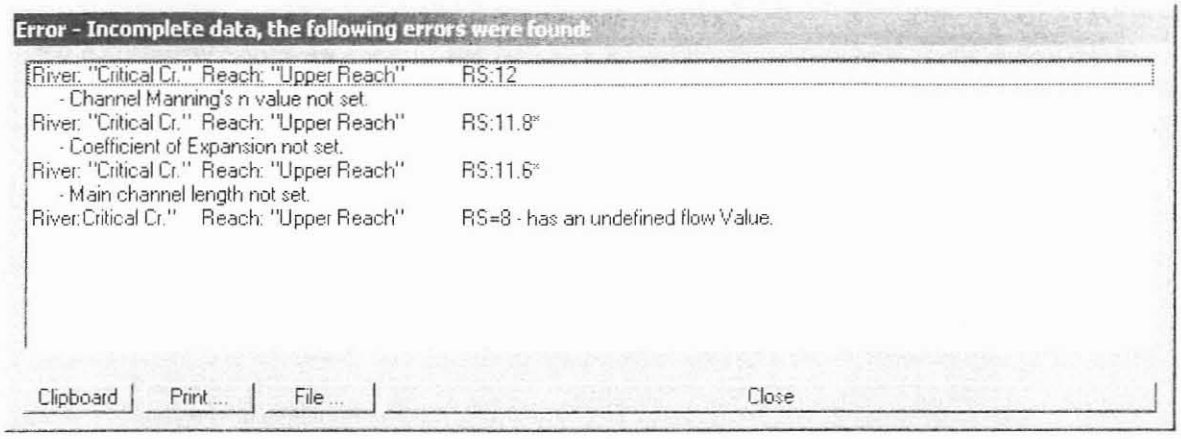


Figure 11.1 Data Completeness Checking Window

Errors, Warnings, and Notes

The HEC-RAS software has a system of Errors, Warnings, and Notes that are passed from the computation programs to the user interface. During the computations, the computation programs will set flags for at a particular node (nodes are cross sections, bridges, culverts, or multiple openings) whenever it is necessary. These message flags are written to the standard output file, along with the computed results for that node. When the user interface reads the computed results from the output file, if any errors, warnings, or notes exist, they are interpreted and displayed in various locations from the interface.

The user can request a summary of all the errors, warnings, and notes that occurred during the computations. This is accomplished by selecting **Summary Errors, Warnings, and Notes** from the **View** menu on the main HEC-RAS window. Once this is selected, a window will pop up displaying all of the messages. The user can select a specific River and Reach, as well as which Profile and Plan to view. The user has the options of expanding the window; printing the messages; or sending them to the windows clipboard. An example of the Errors, Warnings, and Notes window is shown in Figure 11.2.

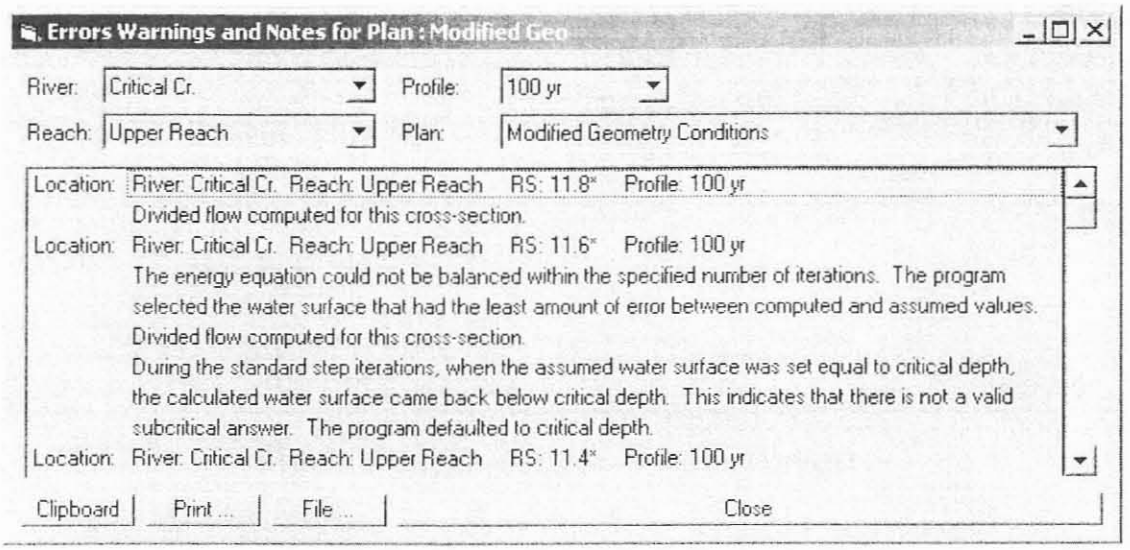


Figure 11.2 Summary of Errors, Warnings, and Notes Window

Besides the summary window, messages will automatically appear on the cross section specific tables. When a cross section or hydraulic structure is being displayed, any errors, warnings, or notes for that location and profile will show up in the Errors, Warnings, and Notes message box at the bottom of the table. An example of this table is shown in Figure 11.3.

In general, the errors, warnings, and notes messages should be self explanatory. The three categories of messages are the following:

ERRORS: Error messages are only sent when there are problems that prevent the program from being able to complete the run.

WARNINGS: Warning messages provide information to the user that may or may not require action on the user's part. In general, whenever a warning is set at a location, the user should review the hydraulic results at that location to ensure that the results are reasonable. If the hydraulic results are found to be reasonable, then the message can be ignored. However, in many instances, a warning level message may require the user to take some action that will cause the message to disappear on future runs. Many of the warning messages are caused by either inadequate or bad data. Some common problems that cause warning messages to occur are the following:

Cross sections spaced to far apart. This can cause several warning messages to be set.

Cross sections starting and ending stations not high enough. If a computed water surface is higher than either end point of the cross section, a warning message will appear.

Bad Starting Water Surface Elevation. If the user specifies a boundary condition that is not possible for the specified flow regime, the program will take action an set an appropriate warning message.

Bad Cross Section Data. This can cause several problems, but most often the program will not be able to balance the energy equation and will default to critical depth.

NOTES: Note level messages are set to provide information to the user about how the program is performing the computations.

Cross Section Output

File Type Options Help

River: Critical Cr. Profile: 100 yr

Reach: Upper Reach RS: 12 Plan: Exist Cond

Plan: Exist Cond Critical Cr. Upper Reach RS: 12 Profile: 100 yr

		Element	Left OB	Channel	Right OB
E.G. Elev (ft)	1815.76				
Vel Head (ft)	0.71	Wt. n-Val.	0.100	0.040	0.100
W.S. Elev (ft)	1815.06	Reach Len. (ft)	500.00	500.00	500.00
Crit W.S. (ft)	1814.46	Flow Area (sq ft)	2137.38	320.82	100.25
E.G. Slope (ft/ft)	0.006851	Area (sq ft)	2137.38	320.82	100.25
Q Total (cfs)	9000.00	Flow (cfs)	5528.62	3370.57	100.81
Top Width (ft)	878.61	Top Width (ft)	698.04	45.00	135.57
Vel Total (ft/s)	3.52	Avg. Vel. (ft/s)	2.59	10.51	1.01
Max Chl Dpth (ft)	11.46	Hydr. Depth (ft)	3.06	7.13	0.74
Conv. Total (cfs)	108731.4	Conv. (cfs)	66792.7	40720.8	1217.9
Length Wtd. (ft)	500.00	Wetted Per. (ft)	700.82	50.80	135.59
Min Ch El (ft)	1803.60	Shear (lb/sq ft)	1.30	2.70	0.32
Alpha	3.67	Stream Power (lb/ft s)	3.37	28.38	0.32
Frctn Loss (ft)	3.82	Cum Volume (acre-ft)	225.37	41.64	10.89
C & E Loss (ft)	0.07	Cum SA (acres)	79.06	6.43	7.75

Errors, Warnings and Notes

The velocity head has changed by more than 0.5 ft (0.15 m). This may indicate the need for additional cross sections.
The energy loss was greater than 1.0 ft (0.3 m). between the current and previous cross section. This may indicate the need for additional cross sections.

Area of left overbank active flow.

Figure 11.3 Cross Section Table with Errors, Warnings, and Notes

Log Output

Steady Flow Log Output

This option allows the user to set the level of the Log file for a steady flow analysis. This file contains information tracing the program process. Log levels can range between 0 and 10, with 0 resulting in no Log output and 10 resulting in the maximum Log output. In general, the Log file output level should not be set unless the user gets an error during the computations. If an error occurs in the computations, set the log file level to an appropriate value. Re-run the computations and then review the log output, try to determine why the program got an error.

When the user selects **Set Log File Output Level** from the **Options** menu, a window will appear as shown in Figure 11.4. The user can set a "Global Log Level," which will be used for all cross sections and every profile. The user can also set log levels at specific locations for specific profiles. In general, it is better to only set the log level at the locations where problems are occurring in the computations. To set the specific location log level, first select the desired reach and river station. Next select the log level and the profile number (the log level can be turned on for all profiles). Once you have everything set, press the **Set** button and the log level will show up in the window below. Log levels can be set at several locations individually. Once all of the Log Levels are set, press the **OK** button to close the window.

Warning !!! - setting the global log output level to 4 or 5 can result in very large log file output. Global log level values of 6 or larger can result in extremely large log files.

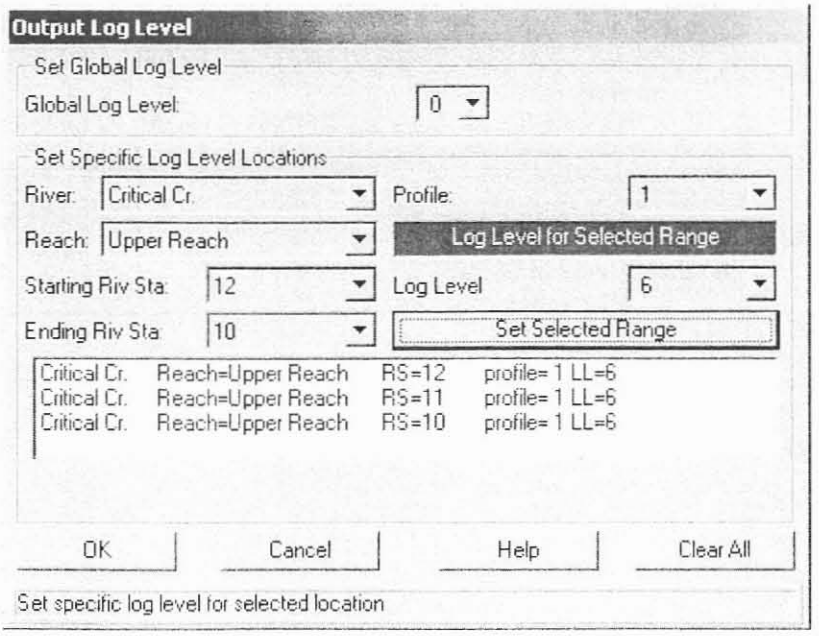


Figure 11.4 Log File Output Level window

Unsteady Flow Log Output

The unsteady flow computation program can write out a detailed log file of its computations. This file is very different from the steady flow program, but serves the purpose of debugging computational problems. This option is turned on by selecting **Output Options** from the **Options** menu on the Unsteady Flow Analysis window. When this option is selected a window will appear as shown in Figure 11.5.

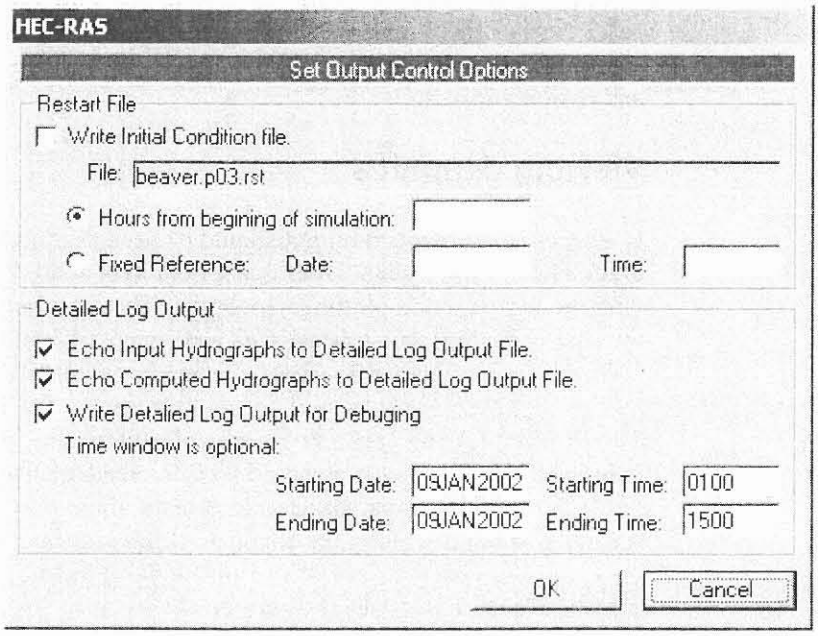


Figure 11.5 Unsteady Flow Output Control Window

As shown in Figure 11.5, this option controls various types of output. To turn on the detailed log output, the user must check the box labeled **Write Detailed Log Output for Debugging**. The user has the option for setting a time window, which will limit the output to only within this time frame. After this option is selected, the computations must be rerun in order for the output to be produced.

Viewing The Log File

This option allows the user to view the contents of the log file. For steady flow analyses, the user brings up the log output by selecting **View Log File** from the **Options** menu of the Steady Flow Analysis window. For unsteady flow analyses, the user brings up the log output by selecting **View Computation Log File** from the **Options** menu of the Unsteady Flow Analysis window. The interface uses the Windows Write program to accomplish viewing the output (unless the user has set a different program to be used as the default file viewer). It is up to the user to set an appropriate font in the Write program. If the user sets a font that uses proportional

spacing, the information in the log file will not line up correctly. Some fonts that work well are: Line Printer; Courier (8 pt.); and Helvetica (8 pt.). Consult your Windows user's manual for information on how to use the Write program.

Reviewing and Debugging the Normal Output

After the user has successfully completed a run, and reviewed all the errors, warnings, and notes, the normal output should be reviewed for consistency and reasonableness.

Viewing Graphics

In general, the graphical output should be used as much as possible to get a quick view of the results. The user should look at all of the **cross sections** with the cross section plotting capability. The cross section plots will assist the user in finding data mistakes, as well as possible modeling mistakes (mistakes in ineffective flow areas, levees, n values, etc.).

The **profile plotting** capability is a good way to get a quick overview of the entire study area. The user should look for sudden changes to the energy grade line and the water surface. In general, these two variables should transition smoothly along the channel. If the user finds rapid changes in the energy or the water surface, the results at those locations should be reviewed closely to ensure that they are correct.

The **X-Y-Z Perspective Plot** can also be used to get a quick view of an entire reach. This plot is very helpful for viewing the top width of the flow area. If the user finds dramatic changes in the top width from one cross section to the next, then the results at those locations should be reviewed closely. Dramatic changes in top width may indicate the need for additional cross sections.

Viewing Tabular Output

There are several types of tabular output. The user should try to make use of all of them when viewing tabular results. In general, the profile summary types of tables should be used to get an overview of some of the key variables at several locations. If any problems are found, or any results that seem suspect, the user should use the detailed output specific tables to get detailed results at a single location.

The Occurrence of Critical Depth

During the steady flow water surface profile calculations, the program may default to critical depth at a cross section in order to continue the calculations. Critical depth can occur for the following reasons:

1. Bad cross section data: If the energy equation can not balance because of bad cross section data, the program defaults to critical depth.

2. Program can not balance the energy equation above or below the top of a levee or ineffective flow area: On occasion, when the program is balancing a water surface that is very close to the top of a levee, or an ineffective flow area, the program may go back and forth (above and below the levee) without being able to balance the energy equation. When this occurs, the program will default to critical depth.

3. Cross sections spaced too far apart: If the cross sections are spaced to far apart, the program may not be able to calculate enough energy losses to obtain a subcritical water surface at the upstream section.

4. Wrong flow regime: When calculating a subcritical profile, and the program comes to a reach that is truly supercritical, the program will default to critical depth. Likewise, when calculating a supercritical profile, if the reach is truly subcritical, the program will default to critical depth.

Computational Program Does Not Run To Completion

While running the computational part of the software, when the steady flow program is finished you should get the message "Finished Steady Flow Simulation" or "Finished Post Processing," for an unsteady flow run. If the user has entered bad data, the computational program may not be able to run to completion. When this happens the program will stop and write an error message to the screen. This message may be a trapped error by the program, or it could be just a generic Fortran error message. Fortran error messages come from the Fortran compiler that was used to develop the computational program. The message basically says that a math error occurred and therefore the program could not continue. When this type of error occurs, it is most often a data input problem. There is a possibility that it could be a bug in the program, but the user should exhaust all the possible data input errors before assuming that the program has a "Bug."

The first step in finding the problem is to realize where the error is occurring. For a steady flow analysis, the program will display which cross section it is working on, and for which profile. This means that the error occurred at that cross section (or hydraulic structure, such as a bridge). Go to the Geometric Data editor and review the input data closely at the problem location.

During an unsteady flow analysis, the program displays the time step that it is working on and the number of iterations it took to solve the equations. As the program is running, if it consistently goes to the maximum number of iterations (20 is the default), the user should take not of the time step that this started to occur. The user must turn on the detailed log output, and then review that output in the vicinity of that particular time step, in order to figure out what is going wrong.

Computational errors often occur at bridges. Check your data closely for any inconsistencies in the bridge geometry. Many of the problems that occur at bridges are due to bad Deck/Roadway data. Go to the Bridge/Culvert Data editor and turn on the option **Highlight Weir, Opening Lid and Ground** from the **View** menu. This option will assist you in finding any geometric mistakes in the bridge data.

CHAPTER 12

Computing Scour at Bridges

The computation of scour at bridges within HEC-RAS is based upon the methods outlined in Hydraulic Engineering Circular No. 18 (FHWA, 2001). Before performing a scour analysis with the HEC-RAS software, the engineer should thoroughly review the procedures outlined in the Hydraulic Engineering Circular No. 18 (HEC 18) report. This chapter presents the data input required for computing contraction scour and local scour at piers and abutments.

For information on the bridge scour equations, please see Chapter 10 of the HEC-RAS Hydraulic Reference Manual.

Contents

- General Modeling Guidelines
- Entering Bridge Scour Data
- Computing Total Bridge Scour

General Modeling Guidelines

In order to perform a bridge scour analysis, the user must first develop a hydraulic model of the river reach containing the bridge to be analyzed. This model should include several cross sections downstream from the bridge, such that any user defined downstream boundary condition does not affect the hydraulic results inside and just upstream of the bridge. The model should also include several cross sections upstream of the bridge, in order to evaluate the long term effects of the bridge on the water surface profile upstream.

The hydraulic modeling of the bridge should be based on the procedures outlined in Chapter 5 of the Hydraulic Reference Manual. If observed data are available, the model should be calibrated to the fullest extent possible. Once the hydraulic model has been calibrated (if observed data are available), the modeler can enter the design events to be used for the scour analysis. In general, the design event for a scour analysis is usually the 100 year (1 percent chance) event. In addition to this event, it is recommended that a 500 year (0.2 percent chance) event also be used in order to evaluate the bridge foundation under a super-flood condition.

The next step is to turn on the flow distribution option in the HEC-RAS software. This option allows for additional output showing the distribution of flow for multiple subdivisions of the left and right overbanks, as well as the main channel. The output of the flow distribution option includes the following items for each flow slice: percentage of flow; flow area; wetted perimeter; conveyance; hydraulic depth; and average velocity. The user can control the number of slices in each flow element (left overbank, main channel, and right overbank), up to a maximum of 45 total slices. The flow distribution output is controlled from the **Options** menu of the **Steady Flow Analysis** window (see Chapter 7, Simulation Options).

The user must request the flow distribution output for the cross sections inside the bridge, the cross section just upstream of the bridge, and the approach section (cross section upstream of the bridge at a distance such that the flow lines are parallel and the flow has not yet begun to contract due to the bridge constriction). Flow distribution output can be requested at additional cross sections, but these are the only cross sections that will be used in the bridge scour computations. The flow distribution option must be turned on in order to get more detailed estimates of the depth and velocity at various locations within the cross section. Once the user has turned this option on, the profile computations must be performed again in order for the flow distribution output to be computed and included in the output file.

After performing the water surface profile calculations for the design events, and computing the flow distribution output, the bridge scour can then be evaluated. The total scour at a highway crossing is comprised of three components: long-term aggradation and degradation; contraction scour; and local scour at piers and abutments. The scour computations in the HEC-RAS software allow the user to compute contraction scour and local scour at piers and abutments. The current version of the HEC-RAS software does not allow

the user to evaluate long-term aggradation and degradation. Long term aggradation and degradation should be evaluated before performing the bridge scour analysis. Procedures for performing this type of analyses are outlined in the HEC No. 18 report.

Entering Bridge Scour Data

The bridge scour computations are performed by opening the **Hydraulic Design Functions** window and selecting the **Scour at Bridges** function. Once this option is selected the program will automatically go to the output file and get the computed output for the approach section, the section just upstream of the bridge, and the sections inside of the bridge. The Hydraulic Design window for Scour at Bridges will appear as shown in Figure 12-1.

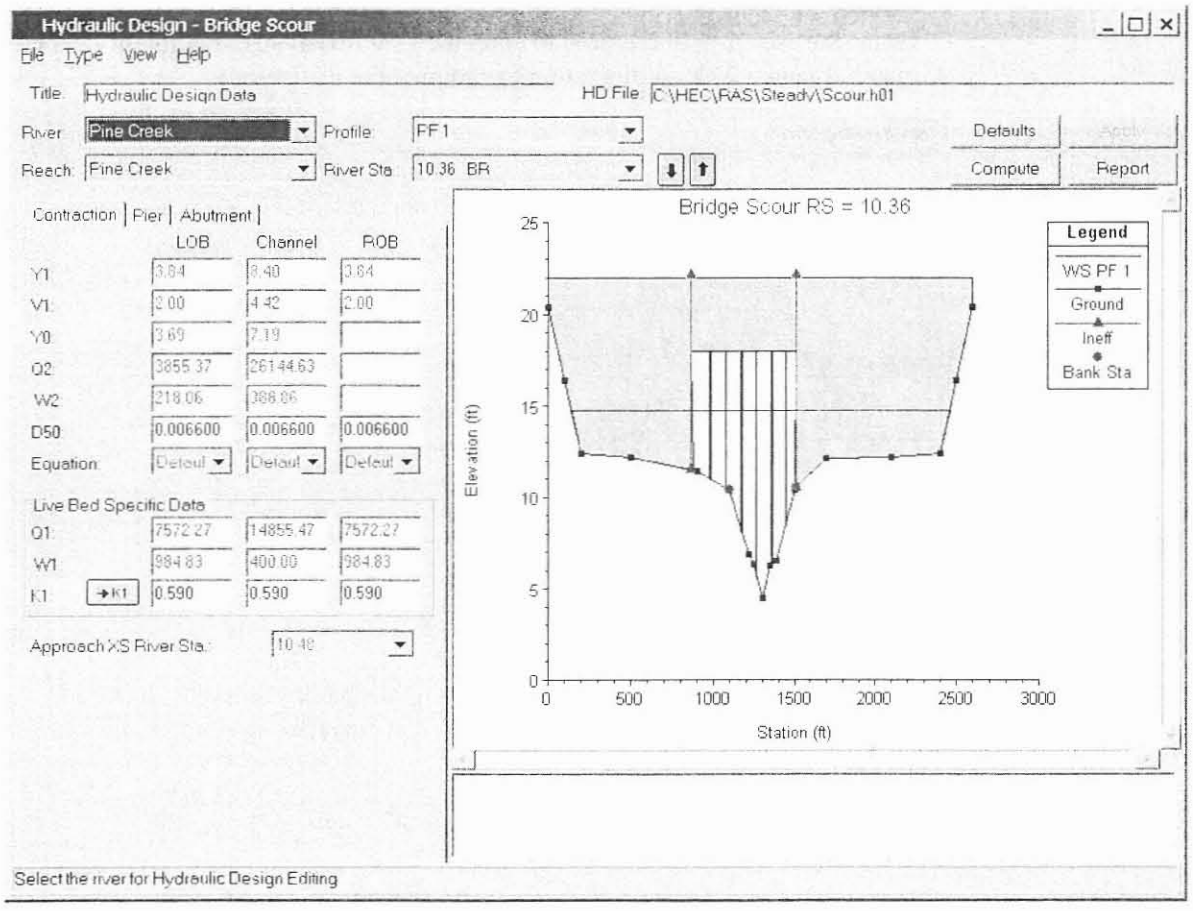


Figure 12.1 Hydraulic Design Window For Scour at Bridges

As shown in Figure 12-1, the Scour at Bridges window contains the input data, a graphic, and a window for summary results. Input data tabs are available for contraction scour, pier scour, and abutment scour. The user is required to enter only a minimal amount of input and the computations can be performed. If the user does not agree with any of the data that the program has selected from the output file, the user can override it by entering their own values. This provides maximum flexibility in using the software.

Entering Contraction Scour Data

Contraction scour can be computed in HEC-RAS by either Laursen's clear-water (Laursen, 1963) or live-bed (Laursen, 1960) contraction scour equations. Figure 12-2 shows all of the data for the contraction scour computations. All of the variables except K1 and D50 are obtained automatically from the HEC-RAS output file. The user can change any variable to whatever value they think is appropriate. To compute contraction scour, the user is only required to enter the D50 (mean size fraction of the bed material) and a water temperature to compute the K1 factor.

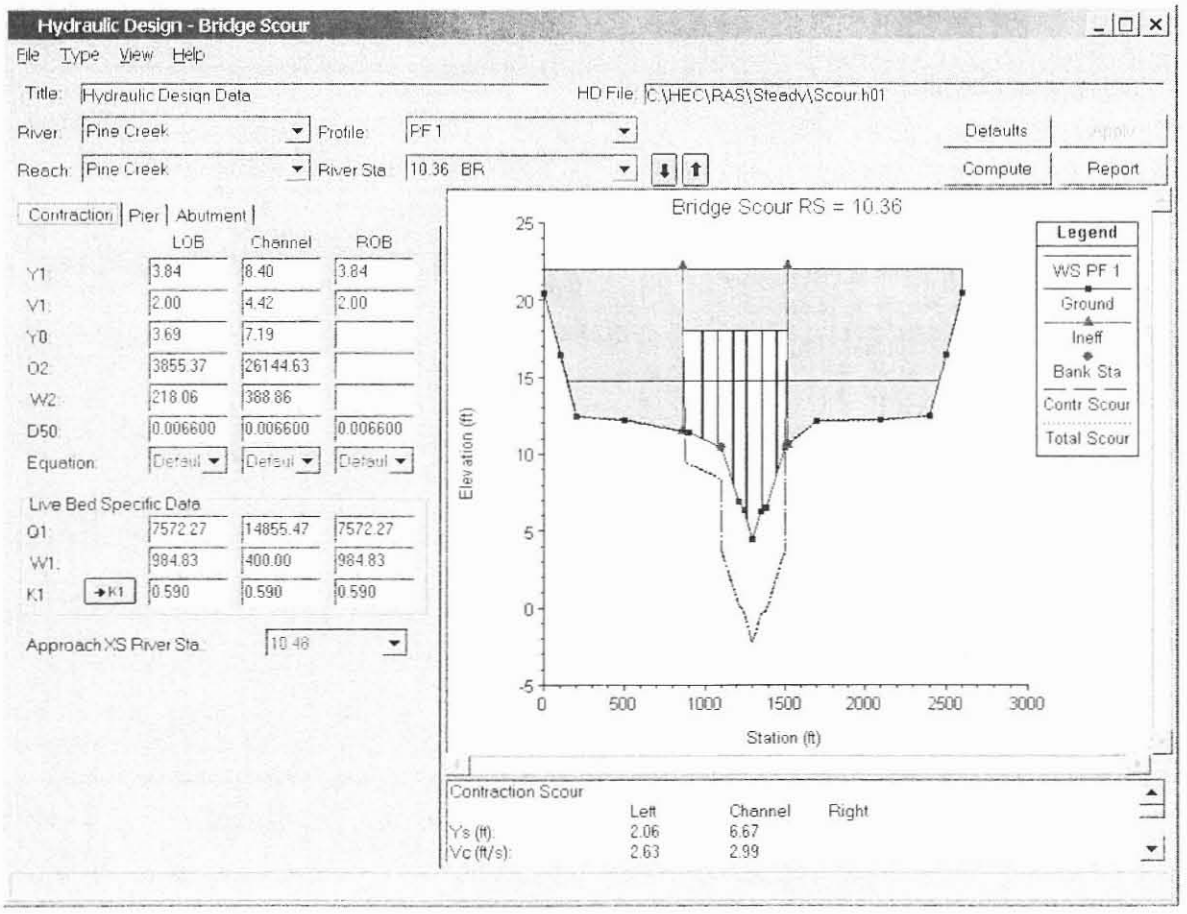


Figure 12.2 Example Contraction Scour Calculation

Each of the variables that are used in the computation of contraction scour are defined below, as well as a description of where each variable is obtained from the output file.

Y1: The average depth (hydraulic depth) in the left overbank, main channel, and the right overbank, at the approach cross-section.

V1: The average velocity of flow in the left overbank, main channel, and right overbank, at the approach section.

Y0: The average depth in the left overbank, main channel, and right overbank, at the contracted section. The contracted section is taken as the cross section inside the bridge at the upstream end of the bridge (section BU).

Q2: The flow in the left overbank, main channel, and right overbank, at the contracted section (section BU).

W2: The top width of the active flow area (not including ineffective flow area), taken at the contracted section (section BU).

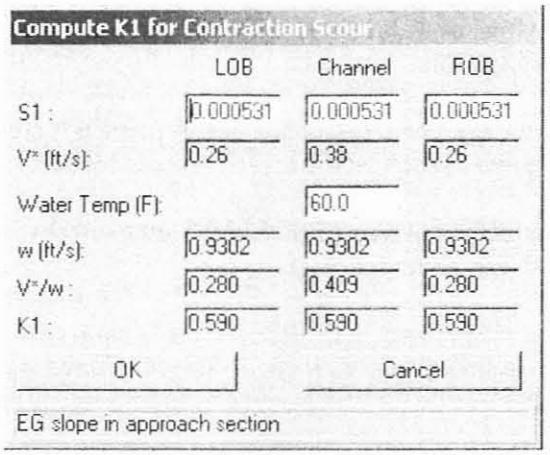
D50: The bed material particle size of which 50% are smaller, for the left overbank, main channel, and the right overbank. These particle sizes must be entered by the user.

Equation: The user has the option to allow the program to decide whether to use the live-bed or clear-water contraction scour equations, or to select a specific equation. If the user selects the **Default** option (program selects which equation is most appropriate), the program must compute V_c, the critical velocity that will transport bed material finer than D50. If the average velocity at the approach cross section is greater than V_c, the program uses the live-bed contraction scour equation. Otherwise, the clear-water contraction scour equation will be used.

Q1: The flow in the left overbank, main channel, and right overbank at the approach cross-section.

W1: The top width of the active flow area (not including ineffective flow area), taken at the approach cross section.

K1: An exponent for the live-bed contraction scour equation that accounts for the mode of bed material transport. The program can compute a value for K1 or the user can enter one. To have the program compute a value, the K1 button must be pressed. Figure 12-3 shows the window that comes up when the K1 button is pressed. Once a water temperature is entered, and the user presses the OK button, the K1 factor will be displayed on the main contraction scour window. K1 is a function of the energy slope (S1) at the approach section, the shear velocity (V^*) at the approach section, water temperature, and the fall velocity (w) of the D50 bed material.



Compute K1 for Contraction Scour

	LOB	Channel	ROB
S1 :	0.000531	0.000531	0.000531
V* (ft/s):	0.26	0.38	0.26
Water Temp (F):		60.0	
w (ft/s):	0.9302	0.9302	0.9302
V*/w :	0.280	0.409	0.280
K1 :	0.590	0.590	0.590

OK Cancel

EG slope in approach section

Figure 12.3 Computation of the K1 Factor

Approach XS River Sta.: The river station of what is being used as the approach cross section. The approach cross section should be located at a point upstream of the bridge just before the flow begins to contract do to the constriction of the bridge opening. The program assumes that the second cross section upstream of the bridge is the approach cross section. If this is not the case, the user can select a different river station to be used as the approach cross section.

As shown in Figure 12-2, the computation of contraction scour is performed separately for the left overbank, main channel, and right overbank. For this example, since there is no right overbank flow inside of the bridge, there is no contraction scour for the right overbank. The summary results show that the computed contraction scour, Y_s, was 2.26 feet (0.69 m) for the left overbank, and 6.67 feet (2.03 m) for the main channel. Also note that the graphic was updated to show how far the bed would be scoured due to the contraction scour.

Entering Pier Scour Data

Pier scour can be computed by either the Colorado State University (CSU) equation (Richardson, et al, 1990) or the Froehlich (1988) equation (the Froehlich equation is not included in the HEC No.18 report). The CSU equation is the default. As shown in Figure 12-4, the user is only required to enter the pier nose shape (K1), the angle of attack for flow hitting the piers, the condition of the bed (K3), and a D95 size fraction for the bed material. All other values are automatically obtained from the HEC-RAS output file.

As shown in Figure 12-4, the user has the option to use the maximum velocity and depth in the main channel, or the local velocity and depth at each pier for the calculation of the pier scour. In general, the maximum velocity and depth are used in order to account for the potential of the main channel thalweg to migrate back and forth within the bridge opening. The migration of the main channel thalweg could cause the maximum potential scour to occur at any one of the bridge piers.

Each of the variables that are used in the computation of pier scour are defined below, as well as a description of where each variable is obtained from the output file.

Maximum V1 Y1: If the user selects this option, the program will find the maximum velocity and depth located in the cross section just upstream and outside of the bridge. The program uses the flow distribution output to obtain these values. The maximum V1 and Y1 will then be used for all of the piers.

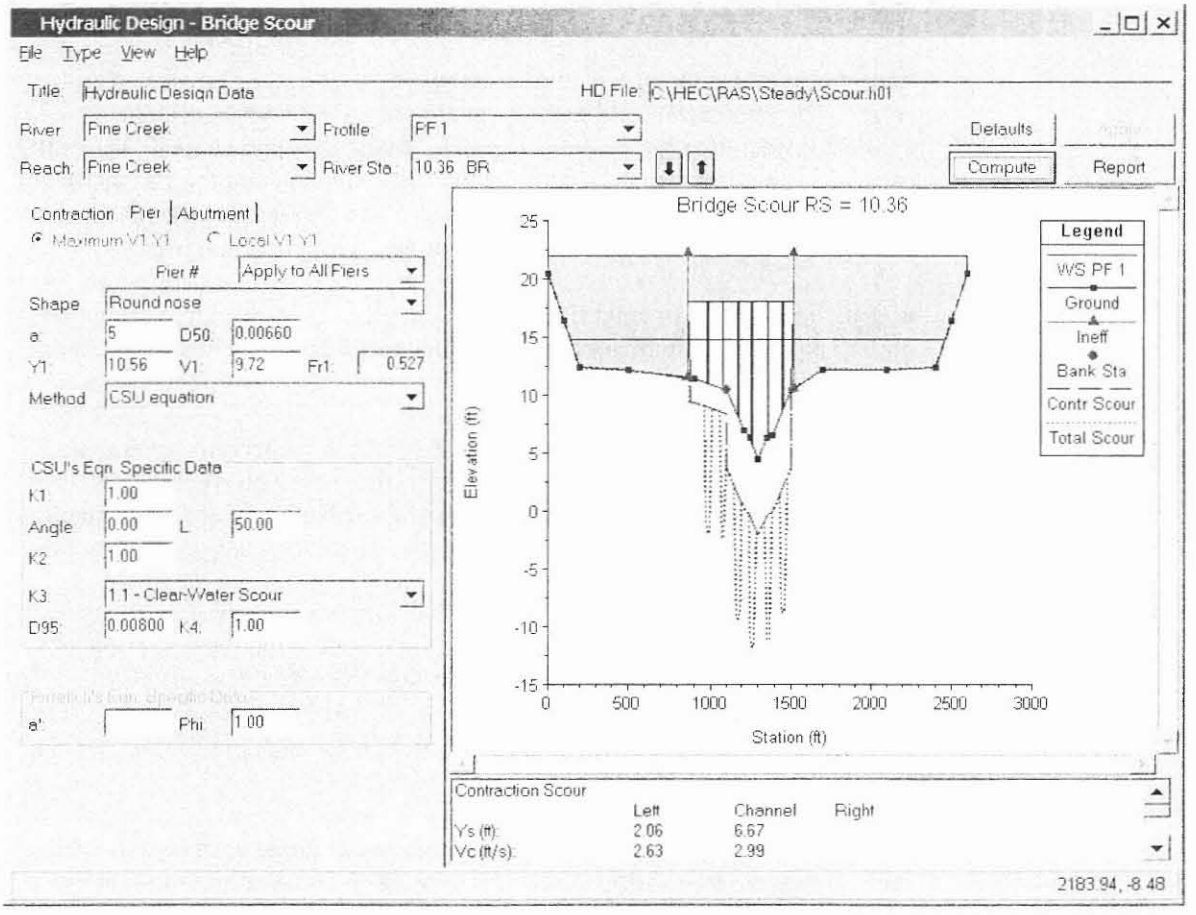


Figure 12.4 Example Pier Scour Computation

Local V1 Y1: If the user selects this option, the program will find the velocity (V1) and depth (Y1) at the cross section just upstream and outside of the bridge that corresponds to the centerline stationing of each of the piers.

Method: The method option allows the user to choose between the CSU equation and the Froehlich equation for the computation of local scour at bridge piers. The CSU equation is the default method.

Pier #: This selection box controls how the data can be entered. When the option "**Apply to All Piers**" is selected, any of the pier data entered by the user will be applied to all of the piers. The user does not have to enter all of the data in this mode, only the portion of the data that should be applied to all of the piers. Optionally, the user can select a specific pier from this selection box. When a specific pier is selected, any data that has already been entered, or is applicable to that pier, will show up in each of the data fields. The user can then enter any missing information for that pier, or change any data that was already set.

Shape: This selection box is used to establish the pier nose (upstream end) shape. The user can select between square nose, round nose, circular cylinder, group of cylinders, or sharp nose (triangular) pier shapes. When the user selects a shape, the K1 factor for the CSU equation and the Phi factor for the Froehlich equation are automatically set. The user can set the pier nose shape for all piers, or a different shape can be entered for each pier.

a: This field is used to enter the width of the pier. The program automatically puts a value in this field based on the bridge input data. The user can change the value.

D50: Median diameter of the bed material of which 50 percent are smaller. This value is automatically filled in for each pier, based on what was entered for the left overbank, main channel, and right overbank, under the contraction scour data. The user can change the value for all piers or any individual pier.

Y1: This field is used to display the depth of water just upstream of each pier. The value is taken from the flow distribution output at the cross section just upstream and outside of the bridge. If the user has selected to use the maximum Y1 and V1 for the pier scour calculations, then this field will show the maximum depth of water in the cross section for each pier. The user can change this value directly for each or all piers.

V1: This field is used to display the average velocity just upstream of each individual pier. The value is taken from the flow distribution output at the cross section just upstream and outside of the bridge. If the user has selected to use the maximum Y1 and V1 for the pier scour calculations, then this field will show the maximum velocity of water in the cross section for all piers. The user can change this value directly for each or all piers.

Angle: This field is used to enter the angle of attack of the flow approaching the pier. If the flow direction upstream of the pier is perpendicular to the pier nose, then the angle would be entered as zero. If the flow is approaching the pier nose at an angle, then that angle should be entered as a positive value in degrees. When an angle is entered, the program automatically sets a value for the K2 coefficient. When the angle is > 5 degrees, K1 is set to 1.0.

L: This field represents the length of the pier through the bridge. The field is automatically set by the program to equal the width of the bridge. The user can change the length for all piers or each individual pier. This length is used in determining the magnitude of the K2 factor.

K1: Correction factor for pier nose shape, used in the CSU equation. This factor is automatically set when the user selects a pier nose shape. The user can override the selected value and enter their own value.

K2: Correction factor for angle of attack of the flow on the pier, used in the CSU equation. This factor is automatically calculated once the user enters the pier width (a), the pier length (L), and the angle of attack (angle).

K3: Correction factor for bed condition, used in the CSU equation. The user can select from: clear-water scour; plane bed and antidune flow; small dunes; medium dunes; and large dunes.

D95: The median size of the bed material of which 95 percent is finer. The D95 size fraction is used in the computation of the K4 factor, and must be entered directly by the user.

K4: The K4 factor is used to decrease scour depths in order to account for armoring of the scour hole. This factor is only applied when the D50 of the bed material is greater than 0.006 feet (0.002 m) and the D95 is greater than 0.06 feet (0.02 m). This factor is automatically calculated by the program, and is a function of D50; D95; a; and the depth of water just upstream of the pier. The K4 factor is used in the CSU equation.

a!: The projected pier width with respect to the direction of the flow. This factor should be calculated by the user and is based on the pier width, shape, angle, and length. This factor is specific to Froehlich's equation.

Phi: Correction factor for pier nose shape, used in the Froehlich equation. This factor is automatically set when the user selects a pier nose shape. The user can override the selected value and enter their own value.

For the example shown in Figure 12-4 the CSU equation was used, resulting in a computed pier scour of 10.85 feet (3.31 m) at each pier (shown under summary results in Figure 12-4). Also shown in Figure 12-4 is an updated graphic with both contraction and pier scour shown.

Entering Abutment Scour Data

Abutment scour can be computed by either the HIRE equation (Richardson, 1990) or Froehlich's equation (Froehlich, 1989). The input data and results for abutment scour computations are shown in Figure 12-5.

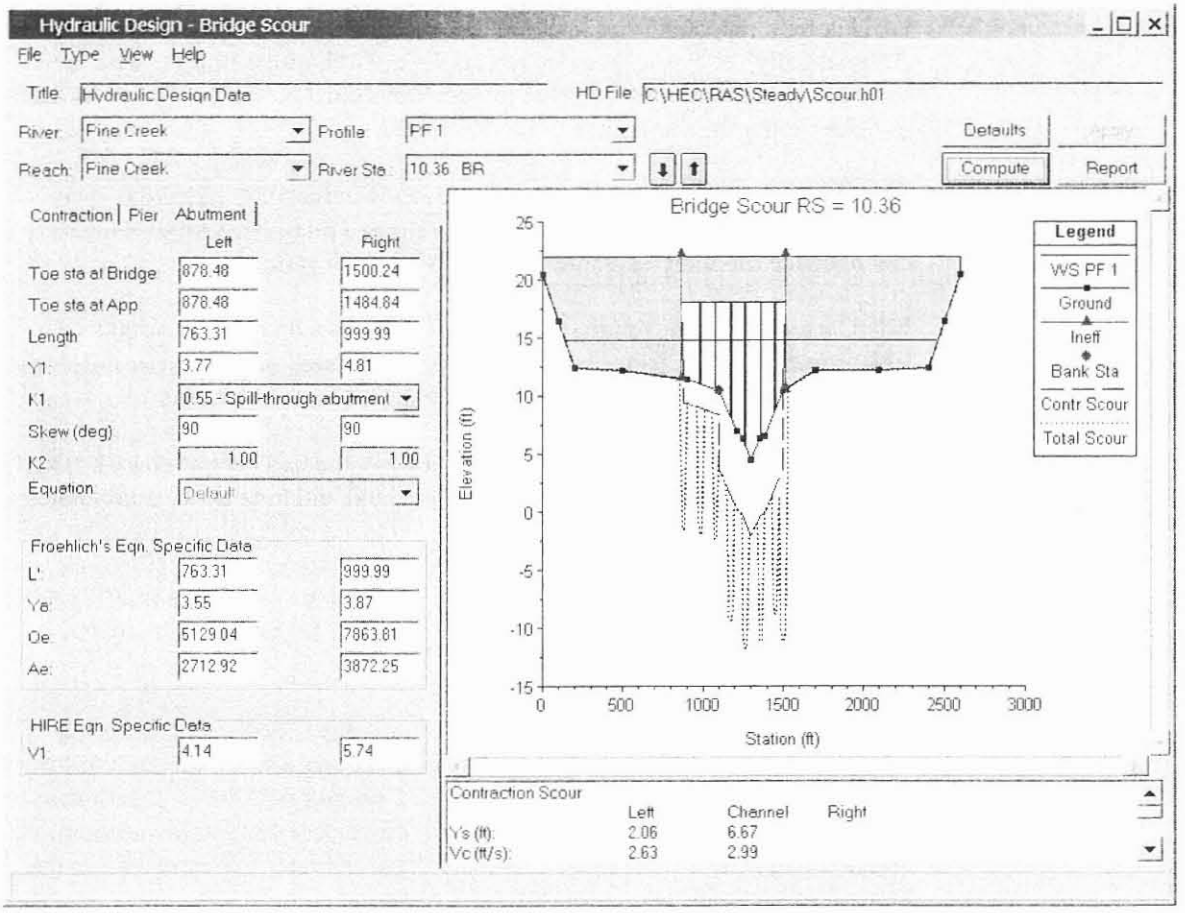


Figure 12.5 Example Abutment Scour Computations

As shown in Figure 12-5, abutment scour is computed separately for the left and right abutment. The user is only required to enter the abutment type (spill-through, vertical, vertical with wing walls). The program automatically selects values for all of the other variables based on the hydraulic output and default settings. However, the user can change any variable. The location of the toe of the abutment is based on where the roadway embankment intersects the natural ground. This stationing is very important because the hydraulic variables used in the abutment scour computations will be obtained from the flow distribution output at this cross section stationing. If the user does not like the stationing that the model picks, they can override it by entering their own value.

Each of the variables that are used in the computation of abutment scour are defined below, as well as a description of where each variable is obtained from the output file.

Toe Sta at Bridge: This field is used to define the stationing in the upstream bridge cross section (section BU), where the toe of the abutment intersects the natural ground. The program automatically selects a value for this stationing at the point where the road embankment and/or abutment data intersects the natural ground cross-section data. The location for the abutment toe stationing can be changed directly in this field.

Toe Sta at App.: This field is used to define the stationing in the approach cross section (section 4), based on projecting the abutment toe station onto the approach cross section. The location for this stationing can be changed directly in this field.

Length: Length of the abutment and road embankment that is obstructing the flow. The program automatically computes this value for both the left and right embankments. The left embankment length is computed as the stationing of the left abutment toe (projected up to the approach cross section) minus the station of the left extent of the active water surface in the approach cross section. The right embankment length is computed as the stationing of the right extent of the active water surface minus the stationing of the toe of the right abutment (projected up to the approach cross section), at the approach cross section. These lengths can be changed directly.

Y1: This value is the computed depth of water at the station of the toe of the embankment, at the cross section just upstream of the bridge. The value is computed by the program as the elevation of the water surface minus the elevation of the ground at the abutment toe stationing. This value can also be changed by the user. This value is used in the HIRE equation.

K1: This value represents a correction factor accounting for abutment shape. The user can choose among: vertical abutments; vertical with wing walls; and spill-through abutments.

Skew: This field is used to enter the angle of attack of the flow against the abutment. A value of 90 degrees should be entered for abutments that are perpendicular to the flow (normal situation). A value less than 90 degrees should be entered if the abutment is pointing in the downstream direction. A value greater than 90 degrees should be entered if the abutments are pointing in the upstream direction. The skew angle is used in computing the K2 factor.

K2: Correction factor for angle of attack of the flow on the abutments. This factor is automatically computed by the program. As the skew angle becomes greater than 90 degrees, this factor increases from a value of one. As the skew angle becomes less than 90 degrees, this value becomes less than one.

Equation: This field allows the user to select a specific equation (either the HIRE or Froehlich equation), or select the default mode. When the default mode is selected, the program will choose the equation that is the most applicable to the situation. The selection is based on computing a factor of the embankment length divided by the approach depth. If this factor is greater that 25, the program will automatically use the HIRE equation. If the factor is equal to or less than 25, the program will automatically use the Froehlich equation.

L!: The length of the abutment (embankment) projected normal to the flow (projected up to the approach cross section). This value is automatically computed by the program once the user enters an abutment length and a skew angle. This value can be changed by the user.

Ya: The average depth of flow (hydraulic depth) that is blocked by the embankment at the approach cross section. This value is computed by projecting the stationing of the abutment toe's up to the approach cross section. From the flow distribution output, the program calculates the area and top width left of the left abutment toe and right of the right abutment toe. Ya is then computed as the area divided by the top width. This value can be changed by the user directly.

Qe: The flow obstructed by the abutment and embankment at the approach cross section. This value is computed by projecting the stationing of the abutment toes onto the approach cross-section. From the flow distribution output, the program calculates the percentage of flow left of the left abutment toe and right of the right abutment toe. These percentages are multiplied by the total flow to obtain the discharge blocked by each embankment. These values can be changed by the user directly.

Ae: The flow area that is obstructed by the abutment and embankment at the approach cross section. This value is computed by projecting the stationing of the abutment toes onto the approach cross-section. From the flow distribution output, the program calculates the area left of the left abutment toe and right of the right abutment toe. These values can be changed by the user directly.

V1: The velocity at the toe of the abutment, taken from the cross section just upstream and outside of the bridge. This velocity is obtained by finding the velocity in the flow distribution output at the corresponding cross section stationing of the abutment toe. These values can be changed by the user directly.

In addition to the abutment input data, once the compute button is pressed, the bridge scour graphic is updated to include the abutment scour and the summary results window displays the computed abutment results. For the example shown in Figure 12-5, the program selected the HIRE equation and computed 10.74 feet (3.27 m) of local scour for the left abutment and 14.90 feet (4.54 m) of local scour for the right abutment.

Computing Total Bridge Scour

The total scour is a combination of the contraction scour and the individual pier and abutment scour at each location. Table 12.1 shows a summary of the computed results, including the total scour.

Table 12.1
Summary of Scour Computations

Contraction Scour

	Left O.B.	Main Channel	Right O.B.
Ys =	2.06 ft (0.63 m)	6.67 ft (2.03 m)	0.00 ft (0.0 m)
Eqn =	Clear-Water	Live-Bed	

Pier Scour

Piers 1-6	Ys = 10.85 ft (3.31 m)
	Eqn. = CSU equation

Abutment Scour

	Left	Right
Ys =	10.74 ft (3.27 m)	14.90 ft (4.54 m)
Eqn =	HIRE equation	HIRE equation

Total Scour

Left Abutment	=	13.00 ft (3.96 m)
Right abutment	=	21.57 ft (6.58 m)
Piers 1-2 (left O.B.)	=	13.11 ft (4.00 m)
Piers 3-6 (main ch.)	=	17.52 ft (5.34 m)

Once all three types of scour data are entered, and the compute button is pressed, the bridge scour graphic is updated to reflect the total computed scour. Shown in Figure 12-6 is the graphic of the final results (the graphic has been zoomed in to see more detail). The graphic and the tabular results can be sent directly to the default printer, or they can be sent to the Windows Clipboard in order to be pasted into a report. A detailed report can be generated, which shows all of the input data, computations, and final results.

The bridge scour input data can be saved by selecting **Save Hydraulic Design Data As** from the **File** menu of the Hydraulic Design Function window. The user is only required to enter a title for the data. The computed bridge scour results are never saved to the hard disk. The computations can be performed in a fraction of a second by simply pressing the compute button. Therefore, when the Hydraulic Design Function window is closed, and later re-opened, the user must press the compute button to get the results.

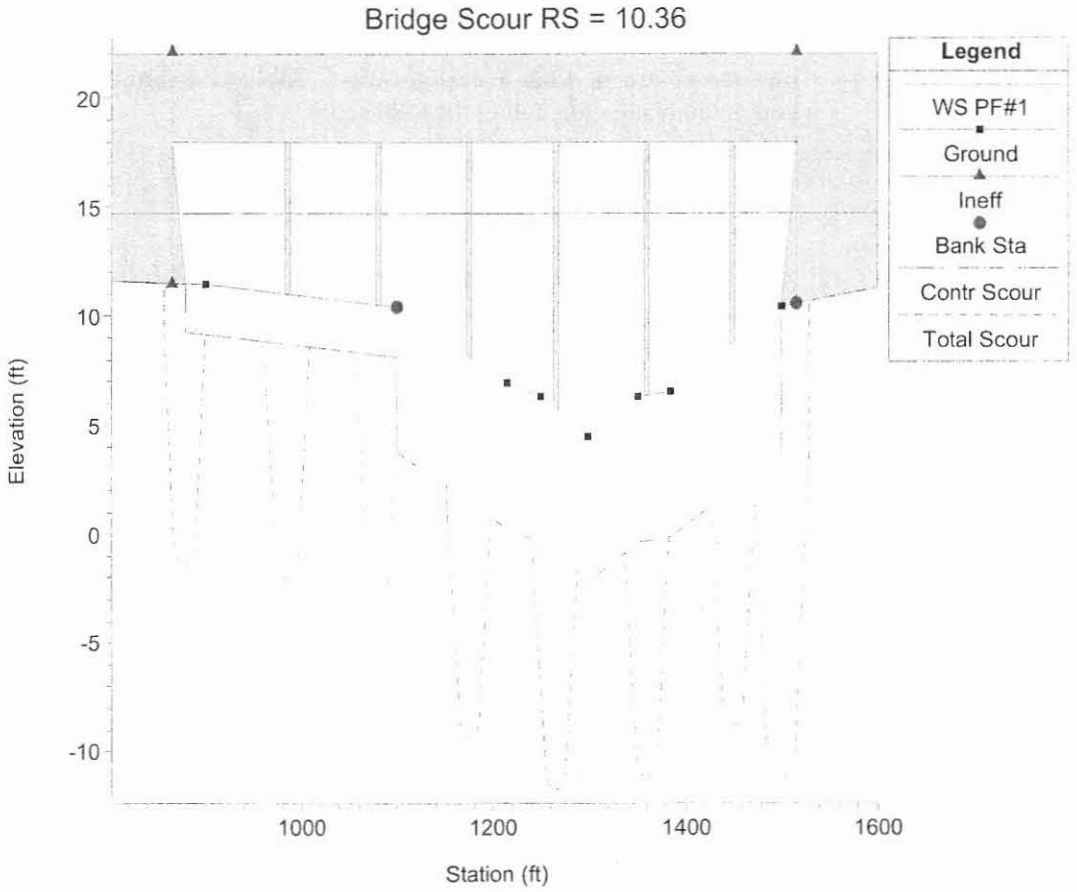


Figure 12-6. Total Scour Depicted in Graphical Form

Miscellaneous

CHAPTER 13

Performing Channel Modifications

The channel modification option in HEC-RAS allows the user to perform a series of trapezoidal cuts into the existing channel geometry. In general, this option is used for planning studies, but it can also be used for hydraulic design of flood control channels.

This chapter does not cover the concepts of stable channel design. This software is designed to evaluate the hydraulics of various channel modifications. It is up to the user to ensure that any channel modification will not cause further scour of the channel bed and banks. Discussions on stable channel design can be found in many hydraulic text books, as well the Corps Engineering Manual "Hydraulic Design of Flood Control Channels" (USACE, 1991).

This chapter discusses: general modeling guidelines for using the channel modification option; how to enter the necessary input data; performing the channel modifications; and how to compare existing condition and modified condition results.

Contents

- General Modeling Guidelines
- Entering Channel Modification Data
- Performing The Channel Modifications
- Comparing Existing and Modified Conditions

General Modeling Guidelines

In order to perform a channel modification analysis, the user should first develop a hydraulic model of the existing river reach containing the area in which the channel modification will be analyzed. This model should include several cross sections downstream from the study reach, such that any user defined downstream boundary condition does not affect the hydraulic results inside the channel modification region. The model should also include several cross sections upstream of the study reach, in order to evaluate the effects of the channel modification on the water surface profile upstream.

Once a model of the existing river system is completed, the user can use the Channel Modification option to perform trapezoidal cuts and fills into the existing geometry. Once the user has performed all of the desired channel modifications, then the modified geometry data is saved into a new geometry file. The user can then create a new plan, which contains the modified geometry and the original flow data that was used under the existing conditions plan. Computations can then be performed for the modified condition, and the user can compare the water surface profiles for both existing and modified conditions.

The channel modification option in HEC-RAS allows for:

- Multiple trapezoidal cuts (up to three)
- Independent specification of left and right trapezoidal side slopes
- Ability to change the Manning's n value for the trapezoidal cut
- Separate bottom widths for each trapezoidal cut
- Ability to set new channel reach lengths
- Multiple ways of locating the main channel centerline
- User can explicitly define the elevation of the new channel invert, or it can be based on the original channel invert, or it can be based on projecting a slope from a downstream cross section or an upstream cross section
- The centerline of the trapezoidal cut can be entered directly, or it can be located midway between the original main channel bank stations
- Option to fill the existing channel before performing cuts
- Cut and fill areas and volumes are computed

Entering Channel Modification Data

Within HEC-RAS, the data for performing a channel modification analysis are entered from the Geometric Data window. The channel modification data are stored within the geometry file of the base geometric data (the geometric data set in which the channel modification is being performed on).

To bring up the Channel Modification Data window, select **Channel Modification** from the **Tools** menu of the Geometric Data window. When this option is selected, a Channel Modification window will appear as shown in Figure 13.1 (except yours will not have any data in it the first time you bring it up).

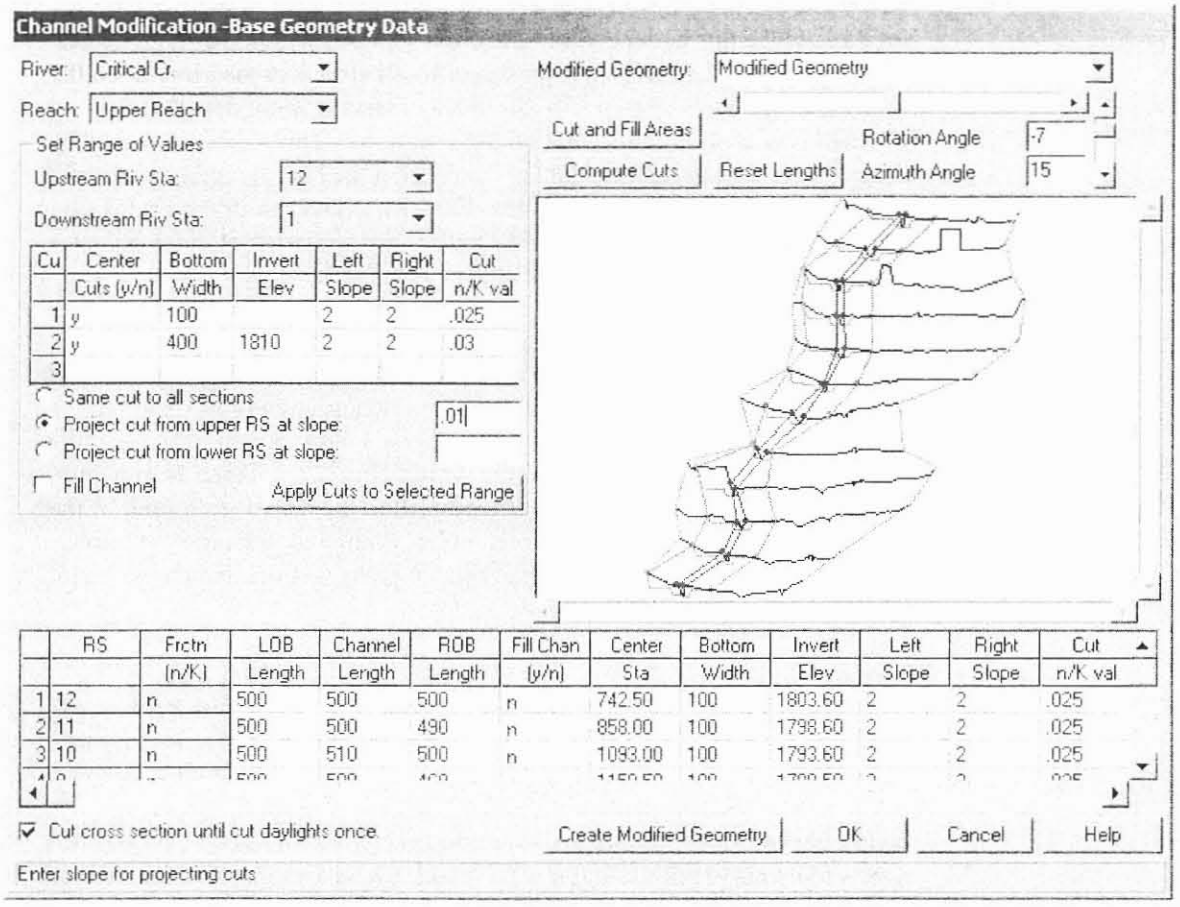


Figure 13.1 Channel Modification Data Editor

As shown in Figure 13.1, there are several pieces of data that the user must enter in order to perform a channel modification analysis. The editor is divided into three separate areas. The top portion of the window contains selection boxes for the River and Reach; titles for the base geometry file and the modified geometry file; buttons for performing the cuts and viewing cut and fill volumes; and controls for rotating the graphic. The middle portion of the window contains a data input area for entering channel modification information over a range of cross sections, as well as a graphic of the cross sections that are being modified. The bottom portion of the window contains a table that lists the channel modification data for all of the cross sections in the selected Reach of a particular River.

The first step in performing a channel modification is to select the River and Reach in which you want to perform the analysis. This is accomplished from the River and Reach selection boxes in the upper left corner of the window. The next step is to select a range of cross sections in which you would like to perform a channel modification. This is accomplished by first selecting a cross section from the **Starting Riv Sta** box and then from the **Ending Riv Sta** box. Once this is done, all of the cross sections within the range of the specified starting and ending river stations will appear in the graphic on the right. The next step is to specify the channel modification data that you would like to apply to this range of cross sections. This is accomplished by entering information into the table contained in the "Set Range of Values" area of the window. This table allows the user to enter information for up to three cuts, which can then be applied to the selected range of cross sections. The information contained in this table is as follows:

Center Cuts (y/n): This column in the table is used to define how the trapezoidal cuts will be centered within the existing cross section data. If the user enters a "y" in this column, then that particular cut will be centered between the existing cross-section main channel bank stations. When all of the cut information is entered, and the **Apply Cuts to Selected Range** button is pressed, the program will automatically fill in the center stationing of the trapezoidal cuts in the lower table. If an "n" is entered, then it is up to the user to specify the center stationing for each cross section, and each cut, in the table at the bottom of the window.

Bottom Width: This column is used for entering the bottom width of the trapezoidal cuts. If this column is left blank, it is assumed that the bottom width will be zero. The user always has the option of directly entering the bottom width for each cross section in the table at the bottom of the window.

Invert Elevation: This column is used to specify the invert elevation of the trapezoidal cuts. If this column is left blank for a particular cut, then it is assumed that the invert elevation of that trapezoidal cut will be set equal to the invert elevation of the existing channel. If the user wants to have invert elevations that are not equal to the existing channel inverts, then they must enter elevations into this column and select one of the slope projection options below this table. The user has the option to use the specified invert elevations for each of the cross sections in the selected range; or they can enter elevations for the most upstream cross section and have the other invert elevations computed by projecting the cuts on a constant slope; or the elevations entered can be applied to the most downstream cross section of the range, and all others will be computed by projecting a user specified slope upstream.

Left Slope: This column is used to specify the slope of the left bank for each of the trapezoidal cuts. The slope is entered in units of horizontal distance to one unit in the vertical. (e.g., a value of 2 means the left bank slope will project 2 feet horizontally for every 1 foot vertically).

Right Slope: This column is used to specify the slope of the right bank for each of the trapezoidal cuts. The slope is entered in units of horizontal distance to one unit in the vertical. (e.g., a value of 2 means the right bank slope will project 2 feet horizontally for every 1 foot vertically).

Cut n Val: This column is used to specify the new Manning's n value to be applied to each of the trapezoidal cuts. If this column is left blank for any cut, then the existing n values will be used for that cut.

Once this table has been filled out, the user must select one of the three slope projection options listed below the table. The three options are:

Same Cut to all sections: If this option is selected, then the channel modification data entered into the table will be applied to all of the cross sections in the selected range.

Project cut from upper RS at slope: When this option is selected, the invert elevations that were entered into the table will be applied to the most upstream cross section in the selected range. The invert elevation of all of the other cross sections will be based on projecting a user entered slope from the most upstream cross section to each cross section downstream. The user must enter a slope when this option is selected. The elevations of each cross sections trapezoidal cuts are based on the user entered slope times the distance that each cross section is from the most upstream cross section. The distance is the cumulative main channel reach length for each of the individual cross sections.

Project cut from lower RS at slope: When this option is selected, the invert elevations that were entered into the table will be applied to the most downstream cross section in the selected range. The invert elevation of all of the other cross sections will be based on projecting a user entered slope from the most downstream cross section to each cross section upstream. The user must enter a slope when this option is selected. The elevations of each cross section's trapezoidal cuts are based on the user entered slope times the distance that each cross section is from the most downstream cross section. The distance is the cumulative main channel reach length for each of the individual cross sections.

A final option that can be applied to the selected range of cross sections is the **Fill Channel** option. When this option is turned on, the main channel of the base cross-section data will be filled before any of the trapezoidal cuts are applied. The main channel is filled to an elevation equal to the elevation of the lower of the two main channel bank stations.

Once the user has filled in all of the desired data in the "Set Range of Values" data area, then the **Apply Cuts to Selected Range** button should be pressed. When this button is pressed, the lower table is filled with the specific information that will be applied to each of the cross sections in the selected range. The cut information is then applied to each of the cross sections, and the graphic is updated to show both the existing cross section and the modified cross sections.

The user has the option of entering and modifying the channel modification data directly in the table at the bottom of the window, or they can use the "Set Range of Values" data area to apply a set of channel cut properties to a range of cross sections (this can be done several times for different ranges of cross sections within the reach).

A final option available to the user is **Cut cross section until cut daylights once**. This is a global option that will be applied to all of the channel modification data. When this option is selected, as the program performs the cutting of the trapezoidal channel, the left and right banks of the channel will start at the bottom of the trapezoid and cut through the ground until they reach open air, then the cutting will stop. If this option is turned off, the left and right banks of the trapezoid will be projected to infinity, continually cutting any ground that lies above them.

Performing the Channel Modifications

Once all of the desired channel modification data are entered for a reach, the user should press the **Compute Cuts** button at the top of the graphic. When this button is pressed, all of the channel modification data from the lower table is applied and the graphic is updated to reflect the new cut information. The user can continue to modify the data and press the **Compute Cuts** button as many times as is necessary to get the desired cuts. The cut information is always applied to the base geometry data.

Once the user has completed the desired channel modifications for the reach, they can view the cut and fill quantities by pressing the **Cut and Fill Areas** button. When this button is pressed, a window will appear as shown in Figure 13.2.

Channel Modification - Cut and Fill Data

River:	Critical Cr.	Reach:	Upper Re						
RS		Area L	Area Ch	Area R	Area T	Volume L	Volume Ch	Volume R	Volume T
		(sq ft)	(sq ft)	(sq ft)	(sq ft)	(cu yd)	(cu yd)	(cu yd)	(cu yd)
3	Cut	594	237	1017	1848	9633	4338	17245	31216
	Fill	0	0	0	0	0	0	0	0
	Net	594	237	1017	1848	9633	4338	17245	31216
2	Cut	406	223	1053	1681	7734	4398	23837	35969
	Fill	0	0	0	0	0	0	0	0
	Net	406	223	1053	1681	7734	4398	23837	35969
1	Cut	429	243	1629	2301	0	0	0	0
	Fill	0	0	0	0	0	0	0	0
	Net	429	243	1629	2301	0	0	0	0
Total	Cut					94647	43670	285801	424118
	Fill					0	0	0	0
	Net					94647	43670	285801	424118

Clipboard | Print ... | File ... | Close

Figure 13.2 Channel Modification Cut and Fill Quantities

The cut and fill quantities table shows the cut, fill, and net areas and volumes for each of the individual cross sections, as well as the totals for the reach. The table shows the cut and fill quantities that were necessary in order to transform the existing cross-section data into the modified cross-section data. The areas and volumes are provided in the categories of left overbank, main channel, right overbank, and total. These categories are based on the main channel bank stations of the base geometry data. The volumes listed at a particular cross section, represent the volume between that cross section and the next downstream cross section. The total volume and area at a particular cross section is the sum of the left overbank, main channel, and right overbank quantities for that individual cross section only. Total volumes for the entire reach are listed at the bottom of the table. The Cut and Fill Quantities table can be printed, sent to a file, or copied to the clipboard, by pressing the desired button at the bottom of the window.

The channel modification option has been set up to work with one Reach of the model at a time. If the user needs to perform channel modifications to more than one reach of a multiple reach model, they can simply select a new reach at any time. While the information in the tables and the graphic only show a single reach, the channel modification information is stored for all of the reaches.

Once the user has finished all of the desired channel modifications, for all of the desired reaches, a new geometry file should be created for the modified geometry. To create a modified geometry file, the user must enter a title for the modified geometry file in the upper right hand side of the window. Once the new geometry file title is entered, the file can be created by pressing the **Create Modified Geometry** button at the bottom of the window. When this button is pressed, a **Save Geometry Data As** window will appear. The user has the options to change the directory in which the geometry file will be stored, change the name of the geometry file title, or select an existing geometry file to over write. Once the user has decided on a title and a directory, the OK button can be pressed to save the modified geometry to the hard disk. However, the original geometry file is still the one that is in memory. If the user wants to work with the new modified geometry file, they will need to open it from the Geometric Data Editor window.

Note: the data entered into the channel modification editor is saved as part of the base geometry file (i.e., it is not saved with the modified geometry file). This allows the user to open the base geometry file and recreate the modified geometry. In order for this data to be saved, the user must select **Save Geometry Data** from the file menu of the geometric data editor, after they have entered the channel modification data.

Comparing Existing and Modified Conditions

Once a modified geometry file is created, the user can create a new plan that will incorporate the modified geometry and the previously defined flow data. This is accomplished by first opening the modified geometry file from the Geometric Data window. The next step is to open the Steady Flow Analysis window and create a new Plan. Creating a plan is accomplished by selecting **New Plan** from the **File** menu of the Steady Flow Analysis window. Once a new plan is created, the computations can be performed.

After the water surface profile computations have been performed for the modified channel conditions, the user can compare the results of the existing and modified conditions on any of the graphics and tables. An example cross-section plot of the two plans is shown in Figure 13.3. Figure 13.3 shows the geometry of the modified and existing conditions, along with the computed water surface elevations from both the existing and modified plans. To display the geometry and results from more than one plan on a graphic, the user can select **Plan** from the **Options** menu on any of the graphics. At the top of the plan selection window, turn on the option that says "**Compare Geometry As Well As Output**." Select the two plans to be viewed and hit the **OK** button. The geometry and output for both plans will be displayed.

In addition to graphical output, the user can review the computed results from both plans in a tabular form. Figure 13.4 shows the computed results for both plans in Standard Table 1 of the Profile Output table.

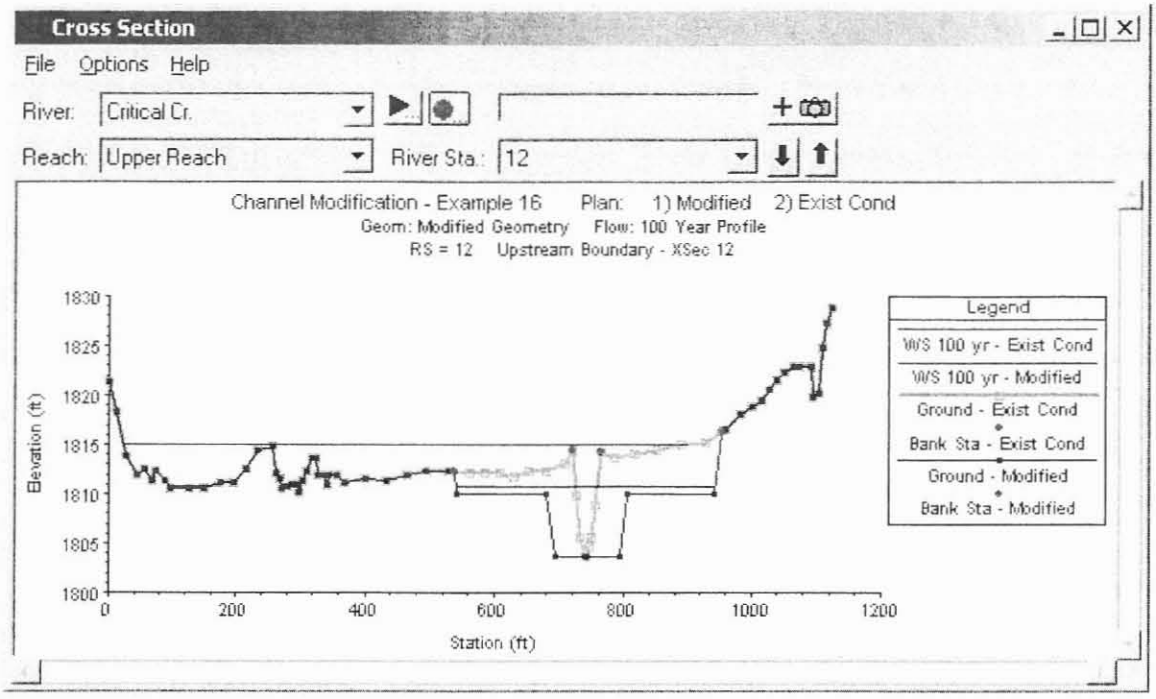


Figure 13.3 Existing and Modified Geometry and Water Surface Elevations

Profile Output Table - Standard Table 1

File Options Std. Tables Help

HEC-RAS River: Critical Cr. Reach: Upper Reach Profile: 100 yr — Reload Data

Reach	River Sta	Plan	Q Total	Min Ch El	W.S. Elev	Crit W.S.	E.G. Elev	E.G. Slope	Vel Chnl	Flow Area
			(cfs)	(ft)	(ft)	(ft)	(ft)	(ft/ft)	(ft/s)	(sq ft)
Upper Reach	12	Exist Cond	9000.00	1803.60	1815.06	1814.46	1815.76	0.006851	10.51	2558.45
Upper Reach	12	Modified	9000.00	1803.60	1810.68	1810.68	1811.95	0.002795	9.06	1000.08
Upper Reach	11	Exist Cond	9000.00	1800.70	1810.42	1810.42	1811.87	0.008552	12.03	1734.74
Upper Reach	11	Modified	9000.00	1798.60	1805.67	1805.67	1806.95	0.002799	9.07	992.61
Upper Reach	10	Exist Cond	9000.00	1794.40	1804.47	1803.69	1804.98	0.010246	10.47	2480.68
Upper Reach	10	Modified	9000.00	1793.60	1800.68	1800.68	1801.89	0.002698	8.90	1172.02
Upper Reach	9	Exist Cond	9000.00	1788.70	1799.31	1799.31	1800.16	0.008851	11.48	2719.81
Upper Reach	9	Modified	9000.00	1788.50	1795.58	1795.58	1796.85	0.002792	9.05	994.18
Upper Reach	8	Exist Cond	9500.00	1784.30	1793.89	1793.89	1795.08	0.008613	12.38	2524.66
Upper Reach	8	Modified	9500.00	1783.50	1790.68	1790.68	1791.99	0.002884	9.16	1061.38

Total flow in cross section.

Figure 13.4 Standard Table 1 With Existing and Modified Conditions

with HEC-RAS

CHAPTER 14

Using GIS Data With HEC-RAS

HEC-RAS has the ability to import three-dimensional (3D) river schematic and cross section data created in a GIS or CADD system. While the HEC-RAS software only utilizes two-dimensional data during the computations, the three-dimensional information is used in the program for display purposes. After the user has completed a hydraulic analysis, the computed water surface profiles can be exported back to the GIS or CADD system for development and display of a flood inundation map.

The importing and exporting of GIS or CADD data is accomplished through the use of formatted ASCII text files. The text files provide a generic way of exchanging data between GIS/CADD systems and HEC-RAS, without adopting any single GIS/CADD system. **Appendix B of this manual provides a detailed description and examples of the file formats used for importing and exporting GIS or CADD data**.

The HEC has developed an ArcView GIS extension called GeoRAS, that was specifically designed to process geospatial data for use with HEC-RAS. The GeoRAS software allows a user to write geometric data to a file in the required format for HEC-RAS. Additionally, the users can read the HEC-RAS results into GeoRAS and perform the flood inundation mapping. This software is not part of the HEC-RAS program. The software and a user's manual are provided as a separate program to be used with ArcView. Also, the Intergraph Corporation has adding the capability to exchange data with HEC-RAS in their Software package called Storm and Sewer Works (Intergraph, 1999)

This chapter discusses how to import GIS or CADD data into HEC-RAS; what additional information will need to be added to complete the data; and how to export the results back to the GIS or CADD system.

Contents

- General Modeling Guidelines
- Importing GIS or CADD Data Into HEC-RAS
- Completing The Data and Performing The Computations
- Exporting Computed Results To The GIS or CADD

General Modeling Guidelines

The current version of HEC-RAS has the ability to import the following geometric data from a GIS/CADD system:

River System Schematic. The structure of the stream network as represented by a series of interconnected reaches. Each reach is represented as a multi-point line, which is assumed to follow the invert of the main channel. The River and Reach labels, as well as the Junction labels, are also imported from the GIS/CADD.

Cross Section Data. The following cross section data can be imported from a GIS/CADD:

1. River, Reach, and River Station identifiers.

2. Cross Section Cut Lines (X and Y coordinates of the plan-view line that represents the cross section). This is a multi-point line that can have two or more points.

3. The cross section surface line. This line is sent to HEC-RAS as a series of X, Y, Z coordinates for each point in the cross section. HEC-RAS transforms these coordinates into station and elevation points (X and Y) for computational purposes. The first station of the cross section is always set to zero. The true (real world) coordinates of the cross section are recomputed from the cross section cut line for the purposes of displaying the data (3D plot).

4. Cross section main channel bank stations.

5. Downstream reach lengths for the left overbank, main channel, and right overbank.

6. Manning's n values.

7. Levee locations and elevations.

8. Ineffective flow areas.

9. Top of road station and elevation for bridges and culverts.

10. Storage area boundaries and elevation versus volume relationship.

At this time, contraction and expansion coefficients, optional cross section properties (blocked obstructions, etc.), and hydraulic structure data (bridges, culverts, etc..) are not imported from a GIS/CADD system. Many of these variables will be added in future versions of the software.

The general procedure for utilizing GIS/CADD data with HEC-RAS is the following:

1. The first step is to start a New Project. This is accomplished from the **File** menu of the main HEC-RAS window.

2. The next step is to go to the Geometric Data editor and import the GIS/CADD data into HEC-RAS. GIS/CADD data are imported by selecting **Import Geomtric Data,** and then **GIS Format** from the **File** menu on the Geometric Data window. This is assuming that you have already used a GIS system to write the required geometry data into a text file, using the required HEC-RAS format. The format of this file is described in Appendix B of this manual.

3. After the GIS data are imported, the user will need to add any additional geometric data that is needed to represent the physical system.

4. The next step is to perform the water surface profile calculations for the desired flow rates.

5. Once the water surface profiles are calculated, the user can then output the results to a text file using the **Export GIS Data** option from the **File** menu of the main HEC-RAS window.

6. The last step is to import the HEC-RAS results file into the GIS/CADD system and develop the floodplain maps for each of the profiles.

Once the user has a project that is utilizing GIS data, then additional data can be imported directly into an existing HEC-RAS geometry file without starting a new project. This allows the user to go back to the GIS and extract additional cross sections on an as-needed basis. The HEC-RAS program will automatically place the new cross sections into the appropriate River and Reach, based on the identifiers defined for each cross section in the GIS import file.

After the user has performed the hydraulic analyses, the computed water surface profiles information can be written to a text file, which can then be imported into the GIS for development and display of floodplain maps. HEC-RAS exports the cross section Cut Line coordinates (X and Y), as well as the water surface elevation for each profile. This is done for every cross section in the model. Additionally, the program exports a series of bounding polygons (one per river reach) for each computed profile. For information on the HEC-RAS GIS export file format, review the detailed write up found in Appendix B of this manual.

Importing GIS or CADD Data Into HEC-RAS

Within HEC-RAS, GIS data are imported from the Geometric Data Window. To import geometric data from a GIS/CADD system into HEC-RAS, the following steps should be followed:

1. The first step is to extract the necessary geometric information from a GIS/CADD system and write it to a text file in the required HEC-RAS format. As mentioned previously, HEC has developed an ArcView GIS extension called GeoRAS to help you do this. Likewise, the Intergraph Corporation has added this capability to their program called Stream and Storm Works. You have the option of obtaining the GeoRAS software from HEC (for use in ArcView); using the software developed by Intergraph; or developing your own routines to extract this data from the GIS/CADD system of your choice. The file formats for the required text file are outlined in Appendix B of this manual.

2. The next step is to start a new project in HEC-RAS. This is accomplished by selecting the **New Project** option from the **File** menu of the main HEC-RAS window. This option will allow the user to enter a project title and filename.

3. After a new project is started, the user should open the Geometric Data Editor. Once the editor is opened, the user can import GIS/CADD data into HEC-RAS by selecting the **Import Geometry Data - GIS Format** option from the **File** menu of the Geometric Data window. When this option is selected, a window will appear (Figure 14.1) in which the user can select the file that contains the geometry data from the GIS.

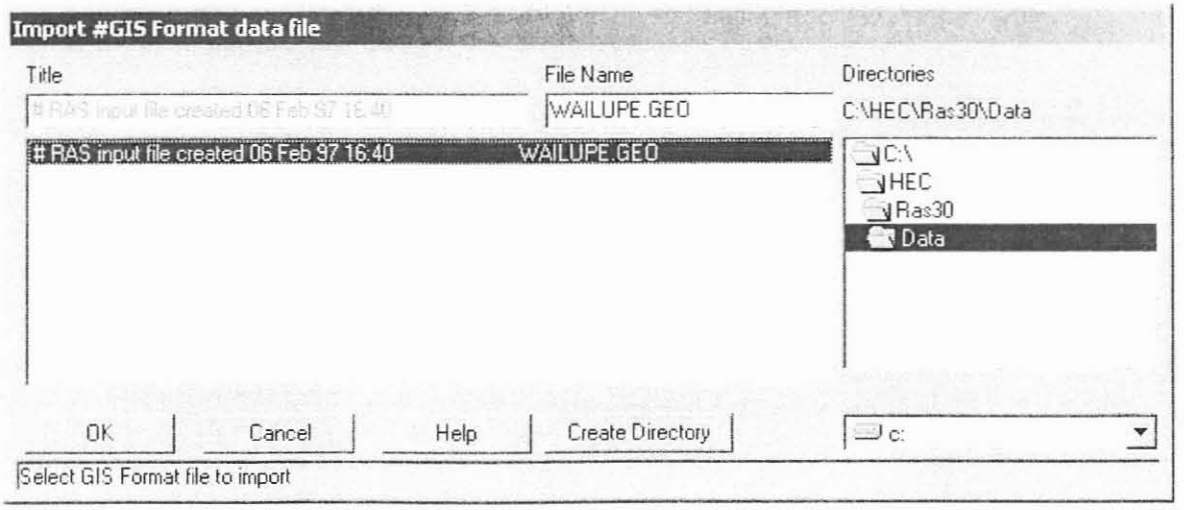


Figure 14.1 Window for Selecting GIS Data File To Import

4. Once the user selects the file containing the GIS data, and then presses the **OK** button, a window will appear that will show you what is available within the import file, and it will allow you to select what you want to import (Figure 14.2)

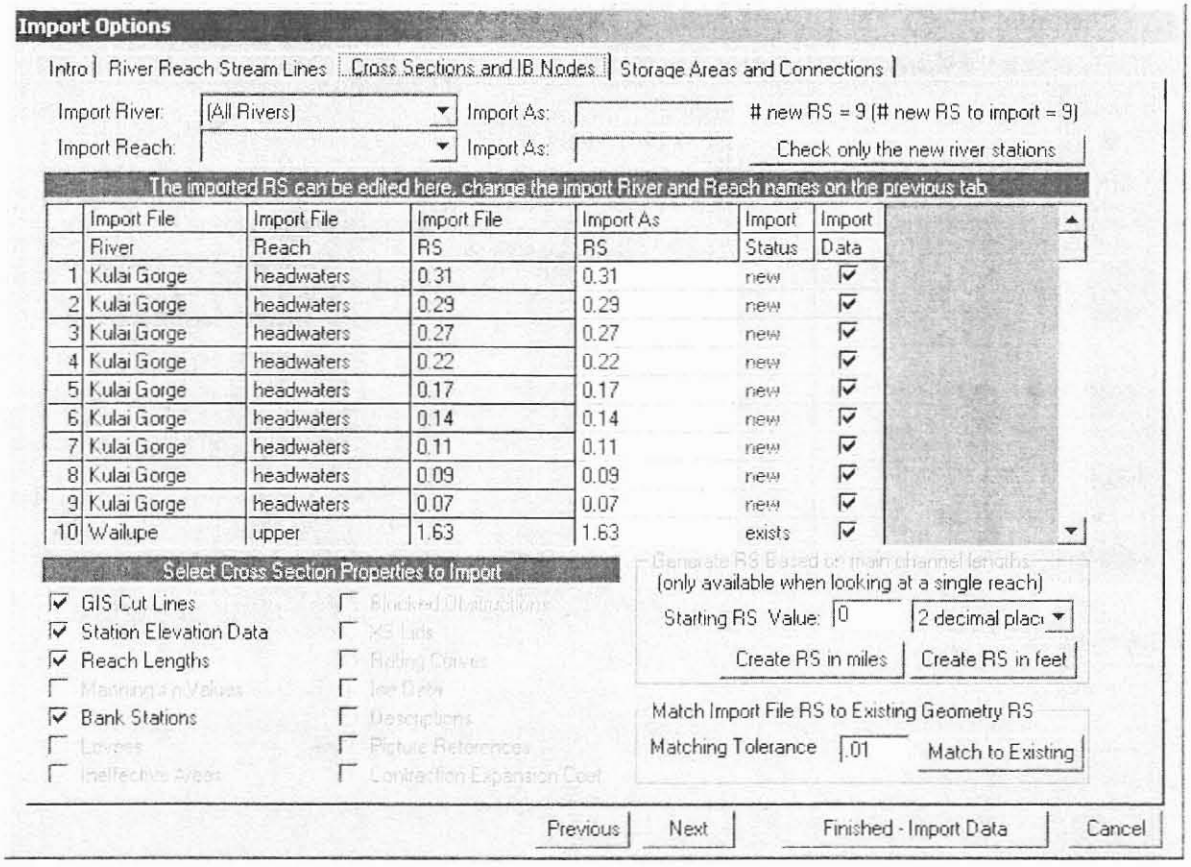


Figure 14.2 GIS Data Import Options Window

As shown in Figure 14.2, the user has several options when importing GIS data. There are four tabs at the top of the window. The first tab is just an introductory tab that explains how to proceed. The second tab **River Reach Steam Lines**, allows the user to select which reaches found in the import file will be imported. The next tab, **Cross Sections and IB Nodes**, allows the user to select which cross sections and internal boundaries (bridges and culverts) will be imported, as well as what specific cross section properties to import. The final tab, **Storage Areas and Connections** allows the user to select which storage areas and connections between storage areas to import.

After making the selections of what to import, The user presses the **Finished – Import** Data button. The data will be imported and a schematic of the river system will show up in the Geometric Data window (Figure 14.3). Once the importing of the data is completed, the user should save the geometric data by selecting **Save Geometry Data As** from the **File** menu of the Geometric Data window.

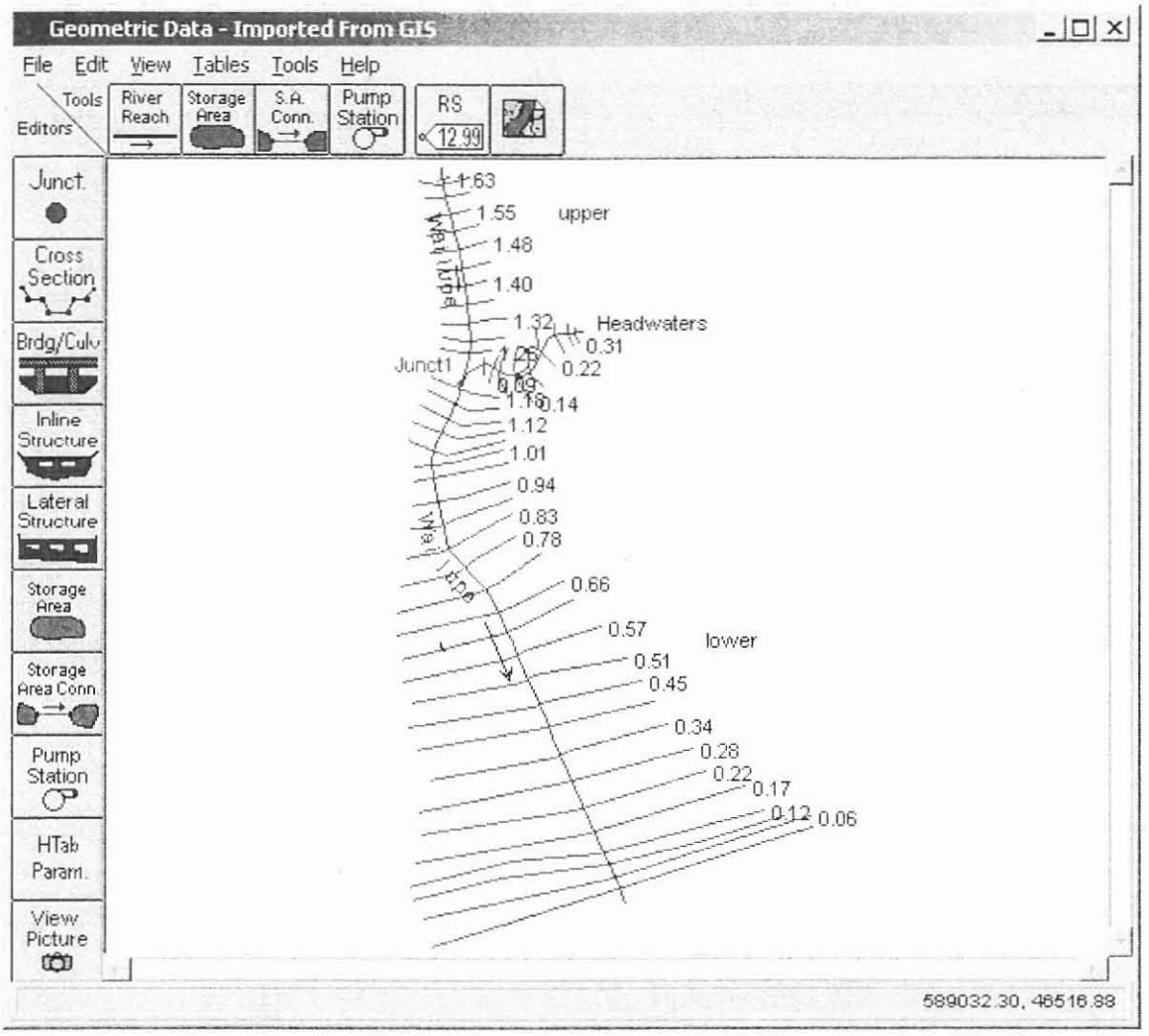


Figure 14.3 River System Schematic of Imported GIS Data

Completing The Data and Performing The Computations

Completing The Geometric Data

Once the user has imported the geometric data from a GIS/CADD system, the next step is to add any additional data required to perform the analyses. Depending on what data was extracted from the GIS (i.e. if n-values were not extracted from the GIS), the user may be required to enter Manning's n values for all of the cross sections. Manning's n values can be entered directly from the cross section data editor (on a cross section by cross section basis) or through the **Manning's n-values** table (this is the preferred way because the n values can be entered more efficiently). The Manning's n value table is available from the **Tables** menu on the Geometric Data window.

In addition to the Manning's n values, the user may need to enter the following data to complete the geometry file: additional user entered cross-section data; interpolated cross sections; optional cross section properties (ineffective flow areas, levees, blocked obstructions, etc.); and hydraulic structures (bridges, culverts, weirs and spillways).

Entering Additional Cross Section Data

If additional cross sections are entered by the user from the cross section editor (or through the HEC-2 import feature), the user will need to enter the coordinates of the cross section strike line in order to maintain a geospatially correct schematic and XYZ plot. The cross section strike line coordinates can be entered by selecting **XS Schematic Lines** from the **Edit** menu on the Geometric Data window. When this option is selected a window will appear as show in Figure 14.4.

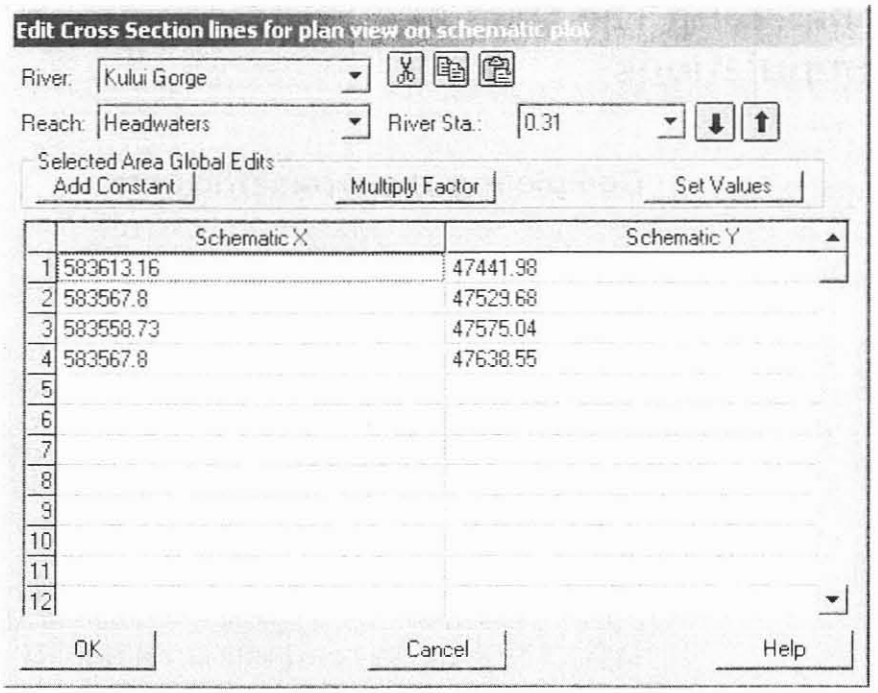


Figure 14.4 Cross Section Schematic Line X and Y Coordinates

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

Performing The Computations and Viewing Results

Once the user has completed the geometric data file, flow data can be entered and the computations can be performed. When utilizing GIS data, there are no special requirements for entering flow data or performing the computations. Once the hydraulic computations are completed, the user can begin to review the output. When GIS data are utilized, the HEC-RAS XYZ perspective plot has the ability to plot a true three dimensional perspective of the river system and the water surface profiles. An example XYZ plot with GIS data is shown in Figure 14.5.

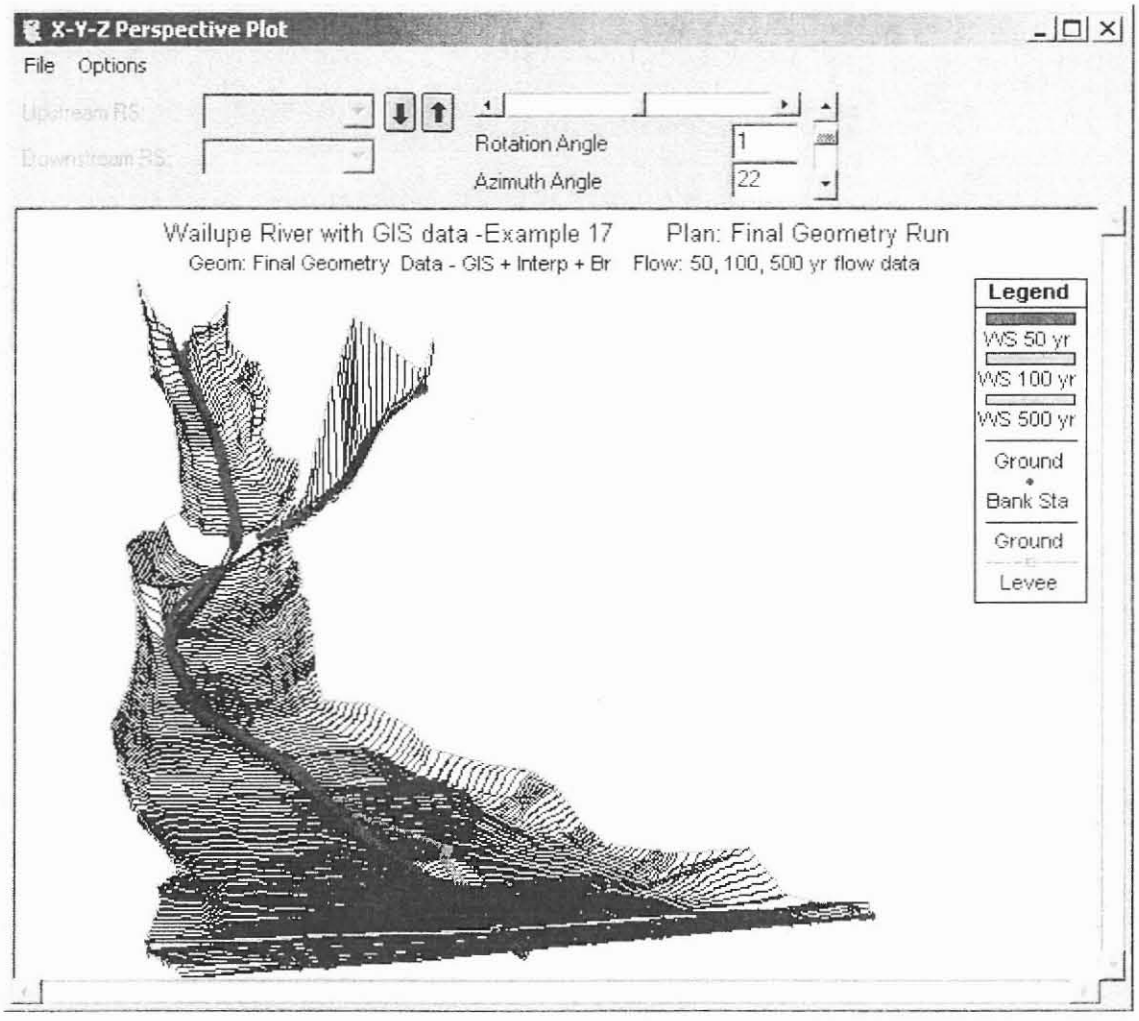


Figure 14.5 XYZ Perspective Plot With GIS Data

Exporting Computed Results To The GIS or CADD

Once the user has completed all of the hydraulic calculations, the computed water surface profiles can be exported to the GIS/CADD in order to develop floodplain maps. The HEC-RAS results are exported to an ASCII text file, which can then be imported by the GIS/CADD system. The format of the HEC-RAS results file is documented in Appendix B of this manual.

Exporting the HEC-RAS computed water surface profiles to a GIS/CADD system is accomplished by selecting **Export GIS Data** from the **File** menu on the main HEC-RAS window. Once this option is selected, a window will appear as shown in Figure 14.6.

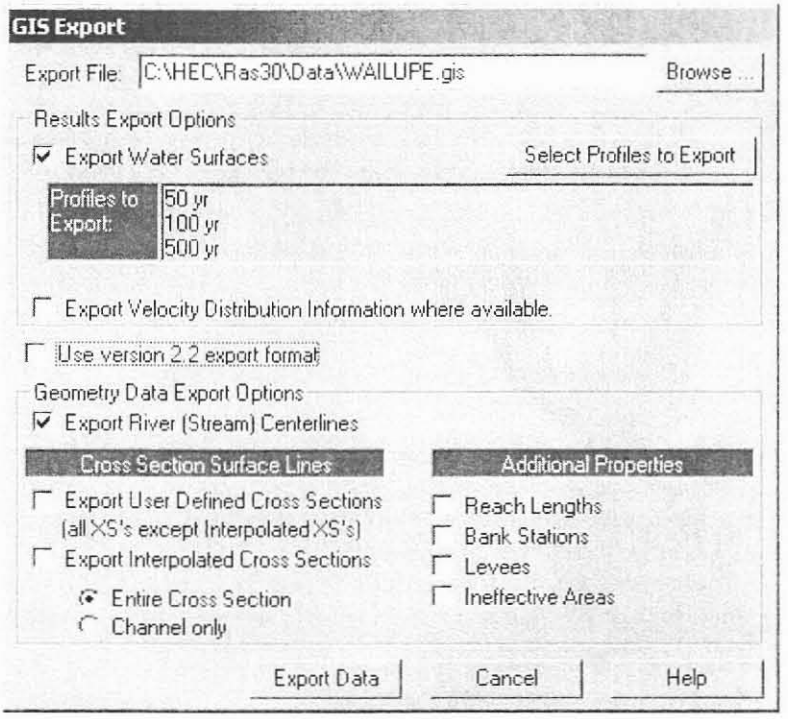


Figure 14.6 Window to Enter a Filename For Exporting Results to GIS

As shown in Figure 14.6, the user first enters a filename for the HEC-RAS Export file. Next, the user can select what they would like to export. Normally the user would select "Export Water Surfaces," and then select which profiles to export by using the **Select Profiles to Export** button. Once these options are selected, the information can be exported by pressing the **Export Data** button.

Additional options are available to export geometry data from HEC-RAS to the GIS/CADD system. This option can be very useful for supplementing terrain data with additional surveyed cross sections. It is a common occurrence for terrain models to have good information in the overbank areas,

but not as good, if at all, in the main channel. HEC-RAS allows the user to export the entire cross section, or just the main channel portion only. Also, the user can send all cross sections, including interpolated sections, or they can turn off the interpolated cross sections. Additionally, there are options to send reach lengths, bank stations, levees, and ineffective flow areas to the GIS system.

In order to use the feature of sending terrain data from HEC-RAS to the GIS, the user must enter geospatial coordinates for all of the cross sections, and the stream centerline before exporting the data. These coordinates are required in order to correctly locate the data spatially within the terrain model.

CHAPTER 15

Stable Channel Design Functions

The channel design functions within HEC-RAS are based upon the methods available in the SAM Hydraulic Design Package for Channels (USACE, 1998), developed by the U.S. Army Corps of Engineers Waterways Experiment Station. This chapter presents the data input required for computing uniform flow parameters, stable channel dimensions, and sediment transport capacity for a given cross section.

For information on the Channel Design Functions equations and theory, please see Chapter 15 of the HEC-RAS Hydraulic Reference Manual.

Contents

- General Modeling Guidelines
- Uniform Flow Computations
- Stable Channel Design
- Sediment Transport Capacity

General Modeling Guidelines

The Stable Channel Design Functions within HEC-RAS are meant to be used as an aid in the design of stable channels. The purpose of this application is to provide the qualitative, easy-to-use methodology of the SAM software package within the HEC-RAS framework. Specifically, the Stable Channel Design Functions will allow the user to easily compute the hydraulic parameters of a given cross section, use that information to design a stable channel with regard to its size and armoring, and determine the sediment transport capacity of that cross section.

General Command Buttons

The general command buttons can be seen in the top right-hand corner of the window shown in figure 15.1. The **Defaults** button restores the current hydraulic design function's fields to the default values. The **Apply** button will store the entries on the current window into memory. These values will remain in memory until a new hydraulic design file is opened or the user exits HEC-RAS. The **Compute** button initiates the computations for whatever hydraulic design function is currently active. The **Report** button displays a printable report providing detailed hydraulic design information. Output will be displayed in the report window if the computations have been run.

Uniform Flow Computations

The uniform flow computations are performed by opening the **Hydraulic Design Functions** window and selecting the **Uniform Flow** from the **Type** menu item.. Once this option is selected the program will automatically go to the geometry file and plot a cross section with the station and elevation data entered into the table. The user can select any cross section from the available rivers and reaches. The Hydraulic Design window for uniform flow will appear as shown in Figure 15.1.

As shown in Figure 15.1, the Uniform Flow window contains the input data, a graphic, and a window for summary results. Input data tabs included are the S/Q/y/n tab and the Width tab. The S/Q/y/n tab is used for calculating the normal slope, discharge, depth, or roughness for the current cross section. The Width tab is used to calculate the bottom width for a uniform flow solution of a user-entered compound channel (with up to 3 trapezoidal templates). The station, elevation, and roughness values for both the current cross section and the user-defined cross section can easily be manipulated in the table and applied to the current geometry file. The user is required to enter only a minimal amount of input and the computations can be performed.

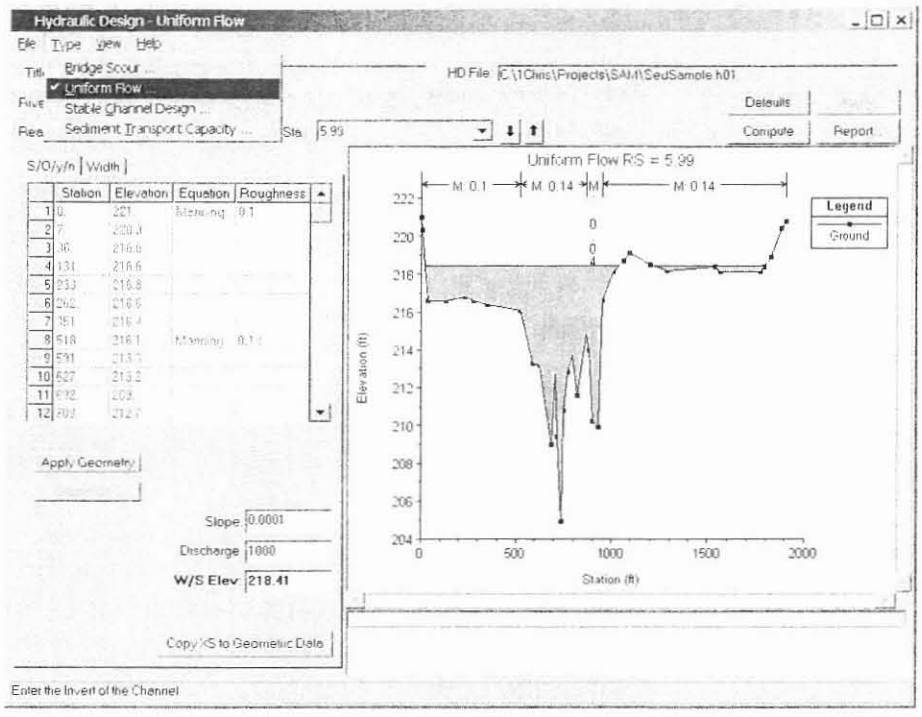


Figure 15.1 Hydraulic Design Window For Uniform Flow

Solving for Slope, Discharge, or W/S Elevation

When the S/Q/y/n tab has been selected, to calculate a slope that satisfies the uniform flow equations for the current cross section, simply enter values into the Discharge and a W/S Elev fields and press the Compute button. A value for the slope is then automatically entered into the Slope field. Likewise, for solving for discharge or water surface elevation, enter values for the other two parameters.

The roughness values are automatically taken from the geometry file, but these can be changed to better represent the bed characteristics of the cross section. In addition to changing the value of the roughness factor (in the default case, Manning's n), the function for defining roughness can be changed. To do this, click on any cell in the equation column of the table and select a function from the dropdown list. The available functions to choose from are Manning's, Keulegan, Strickler, Limerinos, Brownlie, and five grass-lined channel methods. Each of these functions is discussed in detail in Chapter 15 of the Hydraulic Reference Manual.

For the Limerinos and Brownlie functions, gradation distribution is necessary and can be entered by pressing the Gradation button. Only one gradation distribution can be used for a given cross section and should be applied only to the main channel, as these functions were developed for bed material. The Gradation window is shown in Figure 15.2. The following gradation

variables are defined as the following:

d84: The sediment particle size for which 84% of the sediment mixture is finer (mm).

d50: The sediment particle size for which 50% of the sediment mixture is finer (mm).

d16: The sediment particle size for which 16% of the sediment mixture is finer (mm).

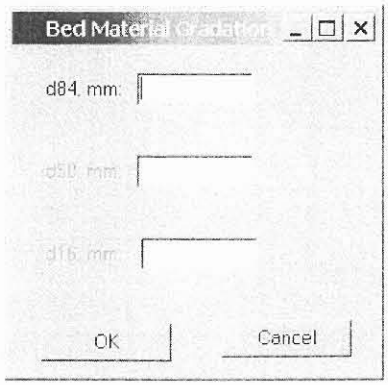


Figure 15.2 Gradation Window

The Brownlie function requires a sediment specific gravity to be entered and the Keulegan function requires a temperature to be entered. The Compute button only becomes active once all required input is entered.

To solve for a roughness value, click on and delete only one of the roughness values in the table. Only one roughness section can be solved for at a time. Make sure Slope, Discharge, and W/S Elev are specified and all other required input are entered. RAS then computes a Manning's n value to satisfy the uniform flow equation for the portion of the cross section that is desired. Then, the roughness value is back-calculated to match the selected roughness function. Only Manning, Keulegan, and Strickler functions can be used to solve for roughness, since the other functions do not have a representative roughness value to solve for.

Once one computation has been made, the value that was solved for will be shown in bold font. For subsequent computations, any of the four uniform flow parameters that is emboldened will be what is solved for to avoid having to delete out the value every time. Once a new parameter is deleted out, it will then be solved for and emboldened.

Solving for Bottom Width

Bottom width can be solved for the uniform flow equation only with a

compound channel that is defined by the user. The compound channel may contain up to three trapezoidal templates, a low flow channel, the main channel, and the overbank channel. The bottom width of either the main channel or the overbank may be solved for. The addition or subtraction of width may be applied to right of centerline, left of centerline or equally to both sides.

When the bottom width tab is selected, the window shown in Figure 15.3 is displayed. To define the compound channel, enter the appropriate values into the compound channel table, which is located below the station elevation table. Data for the Overbank, Main, and Low Flow channels can be entered, however data for the low flow channel can only be applied if a main channel is also defined. The following variables are defined as follows:

SSL: The side slope of the left side of the channel. Entering a value of "0" provides a vertical slope (1Vertical : __Horizontal).

SSR: The side slope of the right side of the channel. Entering a value of "0" provides a vertical slope (1Vertical : __Horizontal).

WL: The bottom width of the left side of the channel from the centerline of the channel to the toe of the side slope (ft or m).

WR: The bottom width of the left side of the channel from the centerline of the channel to the toe of the side slope (ft or m).

Height: The height of the respective channel from its invert to the top of its side slope (ft or m).

Invert: The invert of the respective channel (ft or m).

Once the channel template data is entered, the user may plot the data by selecting Apply Geometry. When this button is selected, the channel design is shown in the plot window and entered in the station elevation table with the default roughness information. A Manning's n value of 0.03 will be applied to each of the channel templates defined. The user may then adjust the roughness values, change the roughness functions, or add more roughness change locations within the cross section on the station elevation table. Any changes made can be reapplied to the plot by pressing Apply Geometry. See Figure 15.4. If either the Brownlie or Limerinos functions are chosen, gradation data will have to be entered.

A value for the energy slope, discharge, and water surface elevation must be entered in the appropriate fields. The user can then select how to solve for the bottom width by using the dropdown boxes in the "Compute Widths" section. Either the main channel or the overbank channel can be solved for and the width can be applied to the left side of the channel (Left of CL only), the right side of the channel (Right of CL only), or equally to both (Total).

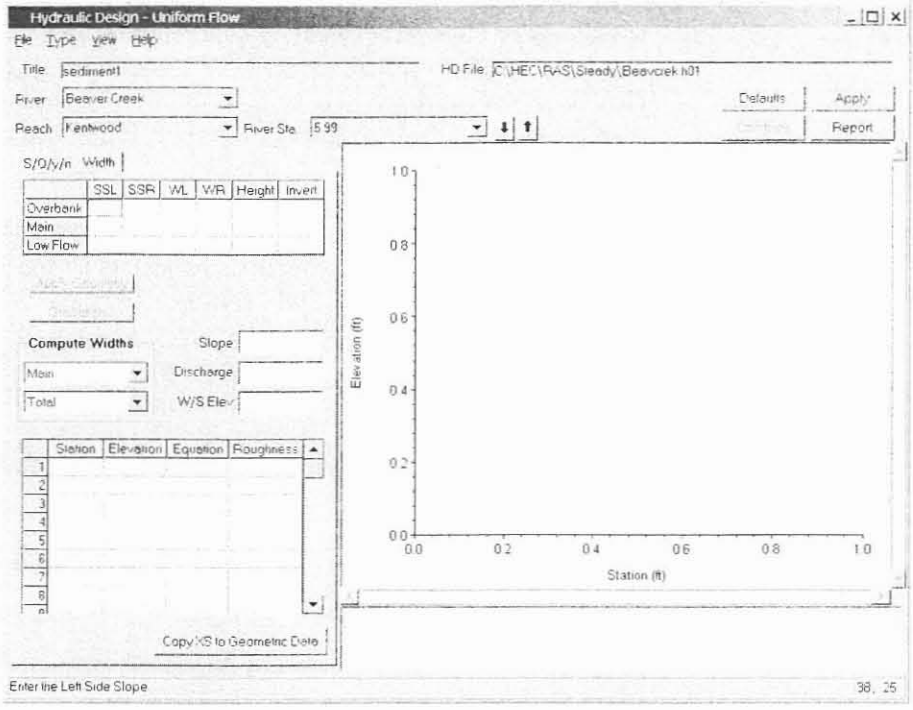


Figure 15.3 Bottom Width Calculation

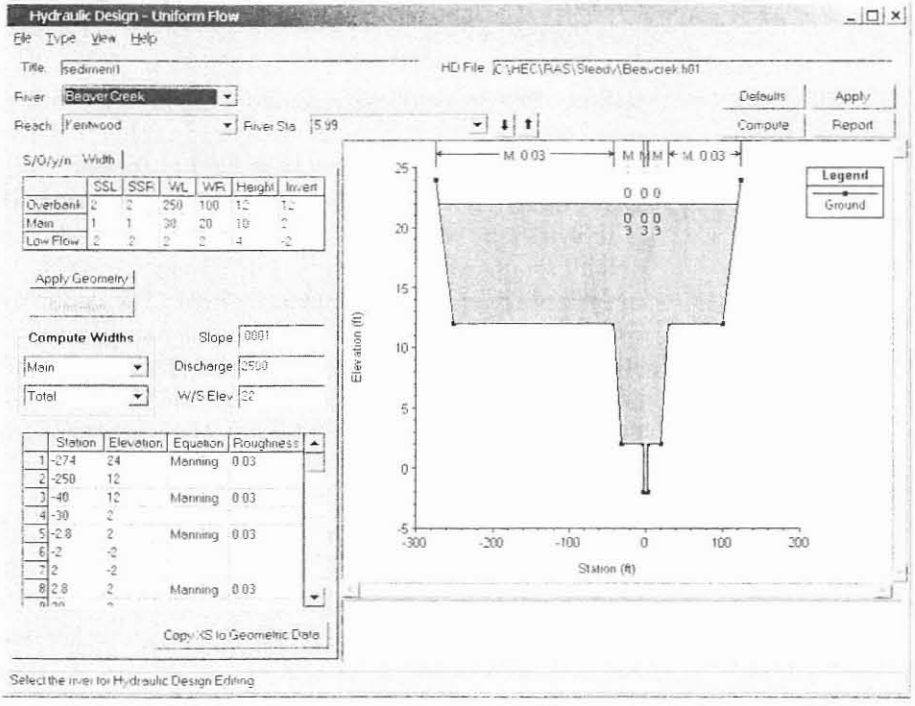


Figure 15.4 Example Bottom Width Data Entry

When all required data is entered, the Compute button will become active. The computations are constrained from creating unrealistic geometries. One example is the overbank bottom width cannot become less than the top width of the main channel. Likewise, the main channel bottom width cannot become less than the low flow channel top width. If this situation occurs within the computations, the user is notified and a course of action is suggested. However, if the top width of a lower channel becomes greater than the bottom width of the channel above it within the calculations, the program automatically increases the upper channel's bottom width to compensate.

When a solution is obtained, the new widths are updated in the compound channel table, the station elevation table and the plot.

Applying Uniform Flow Data to the Geometry File

The resulting cross section, displayed in the plot window can be added to the existing geometry data by clicking on the "Copy XS to Geometric Data" command button. The following window will appear:

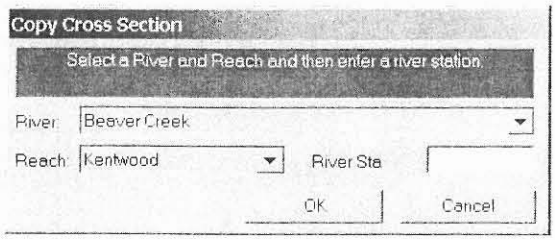


Figure 15.5 Copy Cross Section Window

Enter in the river station you want this cross section to be applied to. If the selected river station already contains a cross section, RAS will ask if you want to copy over it. If there is no cross section at the entered river station, RAS will automatically adjust the distances between the new cross section and its adjacent ones. Make sure that once the new cross section has been copied to the geometry, appropriate values for the bed elevations are reentered. This can easily be done by selecting "Adjust Elevations..." in the Option menu of the Cross Section Data window.

Saving Uniform Flow Data

To save the uniform flow data, click on File...save. This will add all pertinent data from all the HD Functions to an ascii file with the extension *.h##. The content of this file can easily be read within any word processing program.

Stable Channel Design

Stable channels can be computed using three different methods:

- Copeland
- Regime
- Tractive Force

To access the stable channel design window, click on Type…Stable Channel Design in the Hydraulic Design Window. The following window will become active:

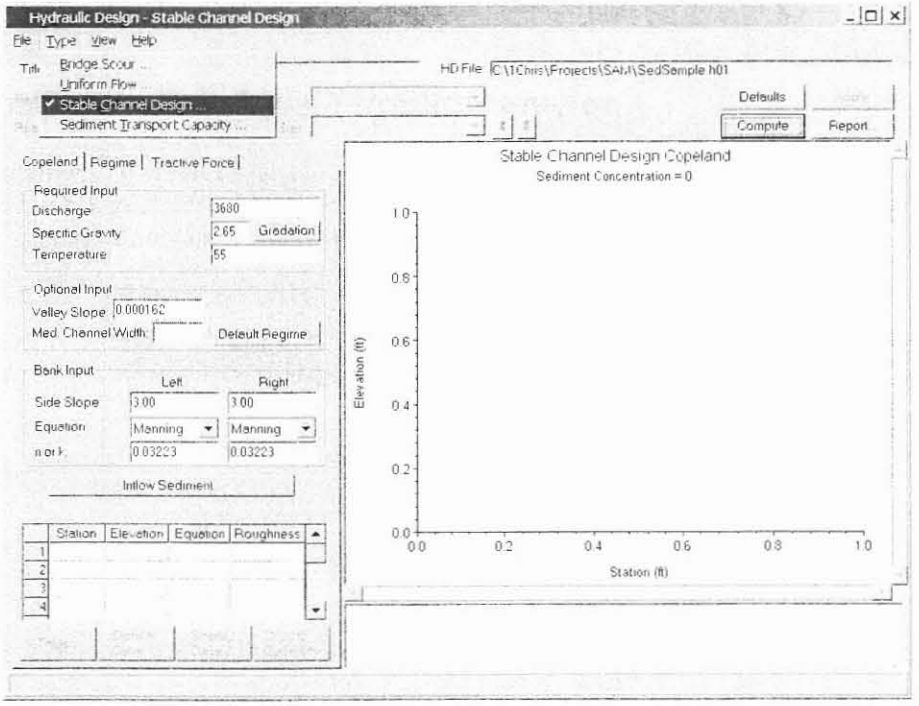


Figure 15.6 Stable Channel Design Window

Copeland Method

To use the Copeland Method, select the tab named "Copeland." There are a number of required and optional fields to enter data into for both the design section and the upstream section. To enter in data for the design section, simply add data to the fields shown.

Discharge: The design discharge. Can be the 2-year, 10-year, bankfull, etc. Must represent the channel forming discharge (cfs or m^3/s).

Specific Gravity: Self-explanatory. Default is 2.65.

Temperature: A representative temperature of the water. Default is 55

degrees F or 10 degrees C.

Valley Slope: (Optional) The maximum possible slope for the channel invert (i.e. no channel sinuosity). If the slope returned is greater than the valley slope, HEC-RAS will indicate that this is a "sediment trap."

Med. Channel Width: (Optional) Median channel width. The median width of the array of 20 bottom widths that are solved for. There will be 9 widths less than and 10 widths greater than the median channel width all at an increment of 0.1 X Med. Channel Width (ft or m). If this is left blank, the median width assigned will be equal to the regime width by the following equation: $B = 2Q^{0.5}$

Side Slope: Slope of the left and right side slopes. (1Vertical : __Horizontal).

Equation: Can choose from Mannings or Strickler to solve for the side slope roughness.

n or k: If Mannings is selected, enter a Mannings "n" value. If Strickler is selected, enter a "k" value (ft or m for k values).

Gradation of the sediment is required for Copeland method and can be entered by clicking on the Gradation command button. Values for d_{84}, d_{50}, and d_{16} must be entered.

The user has the ability to designate the default regime for the computations. The HEC-RAS default is lower regime, but this can be changed by clicking on the "Default Regime..." button and selecting "Upper Regime". Any time the computations result in a solution that is in the transitional regime, the default regime will be used and the user will be notified in the output table that this occurred. See chapter 12 of the Hydraulic Design Manual for more information.

Once all required data for the design section has been entered, click on the "Inflow Sediment..." command button to input information about the upstream section for sediment concentration computations. The window shown in figure 15.7 becomes active. The user can either enter in a value for the inflowing sediment concentration or let HEC-RAS calculate it. If HEC-RAS is to calculate the inflow sediment concentration, then the following information about the upstream section must be entered:

Supply Reach Bottom Width: Width of the bed of the supply reach (ft or m).

Supply Reach Bank Height: A representative value of the bank elevation minus the channel invert elevation of the supply section. This is only used in the computations to target a depth and does not limit the solution to this height (ft or m).

Supply Energy Slope: A representative energy slope at the supply section.

Water surface slope is typically used.

Side Slope: Slope of the left and right side slopes of the supply section. (1Vertical : __Horizontal).

Equation: Can choose from Mannings or Strickler to solve for the side slope roughness of the supply section.

n or k: If Mannings is selected, enter a Mannings "n" value. If Strickler is selected, enter a "k" value for the supply section (ft or m for k values).

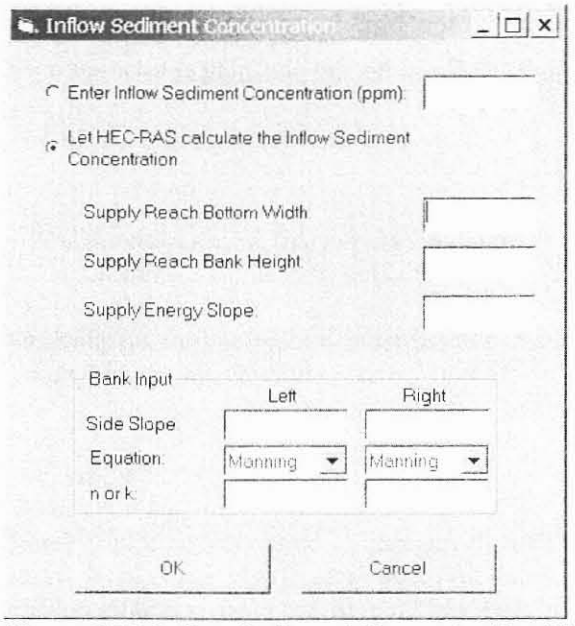


Figure 15.7 Inflow Sediment Concentration Window

Click OK to apply the input and return to the main HD Functions window. Once all of the required input has been entered, the Compute button will be activated. Click the Compute button to run the computations. When the computations are complete, the output table will be shown. The output table lists all of the channel widths solved for along with the corresponding depth, slope, composite n value, hydraulic radius, velocity, Froude number, shear stress and bed transport regime. An example is shown in figure 15.8. There will be twenty different stable channel geometries plus one for the minimum stream power. The user can select one of these geometries for display on the plot window. Once the desired section is selected, click OK and the HD Functions window will become active with the selected section plotted in the plot window.

When the computations have been run, the Table button, the two Stability Curve buttons and the Copy to Geometry button become active. The Table button simply allows the user to pull up the output table again, and select a different stable section, if desired. Clicking on the Stability Curve 1 button will bring up a plot of the stability curve showing slope versus width,

indicating for what slope/width combination aggradation or degradation can be expected. Figure 15.9 shows an example.

Stable Channel Design, Copeland Method

Select a stable channel dimension to display

Sediment Concentration, ppm = 17.47

Bottom Width	Depth	Energy Slope	Composite n-value	Hyd Radius	Velocity	Froude Number	Shear Stress	Bed Regime
12	18.07	0.000176	0.0318	9.85	2.84	0.12	0.21	Lower
24	18.37	0.000127	0.0313	10.32	2.53	0.1	0.15	Lower
36	17.45	0.000107	0.0310	10.53	2.39	0.1	0.12	Lower
49	16.36	0.000095	0.0303	10.52	2.29	0.1	0.1	Lower
61	15.38	0.000088	0.0298	10.41	2.23	0.1	0.08	Lower
73	14.46	0.000084	0.0294	10.23	2.19	0.1	0.08	Lower
85	13.61	0.000082	0.0290	10.00	2.15	0.1	0.07	Lower
97	12.83	0.00008	0.0287	9.76	2.12	0.1	0.06	Lower
109	12.12	0.000079	0.0283	9.49	2.09	0.11	0.06	Lower
121	11.47	0.000078	0.0279	9.20	2.06	0.11	0.06	Lower
133	10.89	0.000078	0.0276	8.93	2.04	0.11	0.05	Lower
146	10.32	0.000078	0.0274	8.64	2.02	0.11	0.05	Lower
158	9.83	0.000078	0.0272	8.37	2	0.11	0.05	Lower
170	9.4	0.000079	0.0269	8.11	1.98	0.11	0.05	Lower
182	8.99	0.000079	0.0267	7.86	1.96	0.12	0.04	Lower
194	8.63	0.00008	0.0265	7.63	1.94	0.12	0.04	Lower
206	8.29	0.000081	0.0264	7.40	1.92	0.12	0.04	Lower
218	7.97	0.000082	0.0262	7.18	1.91	0.12	0.04	Lower
231	7.66	0.000083	0.0260	6.95	1.89	0.12	0.04	Lower
243	7.4	0.000084	0.0259	6.76	1.88	0.12	0.04	Lower
------	Minimum	Stream	Power	------				
133.68	10.86	0.000078	0.02781	8.95	2.04	0.11	0.05	Lower

Red Color indicates that the computed slope is greater than the user-entered valley slope, indicating a potential sediment trap

OK | Cancel

* Indicates transitional regime. The default regime was used for the computations

Figure 15.8 Copeland Method Output Table

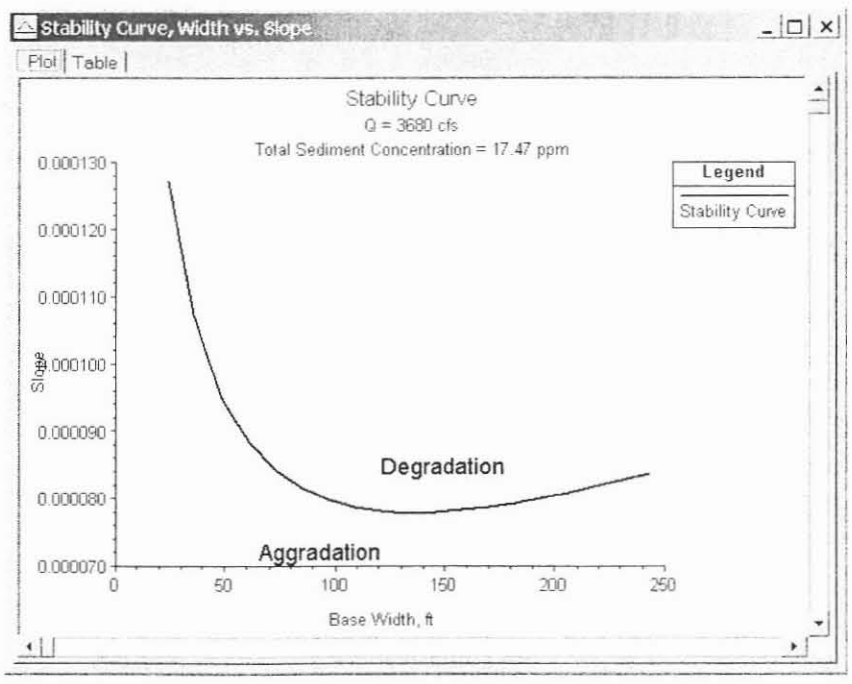


Figure 15.9 Stability Curve

Stability Curve 2 brings up a similar plot, only with slope compared to *depth*. In addition to viewing the plots, the table tab can be clicked to view the stability curves in tabular form.

As with the uniform flow computations, the section that has been plotted from the Copeland Method can be applied to the current geometry file by clicking on the Copy to Geometry button.

Regime Method

To use the Regime Method, select the tab named "Regime." The window shown in figure 15.10 becomes active.

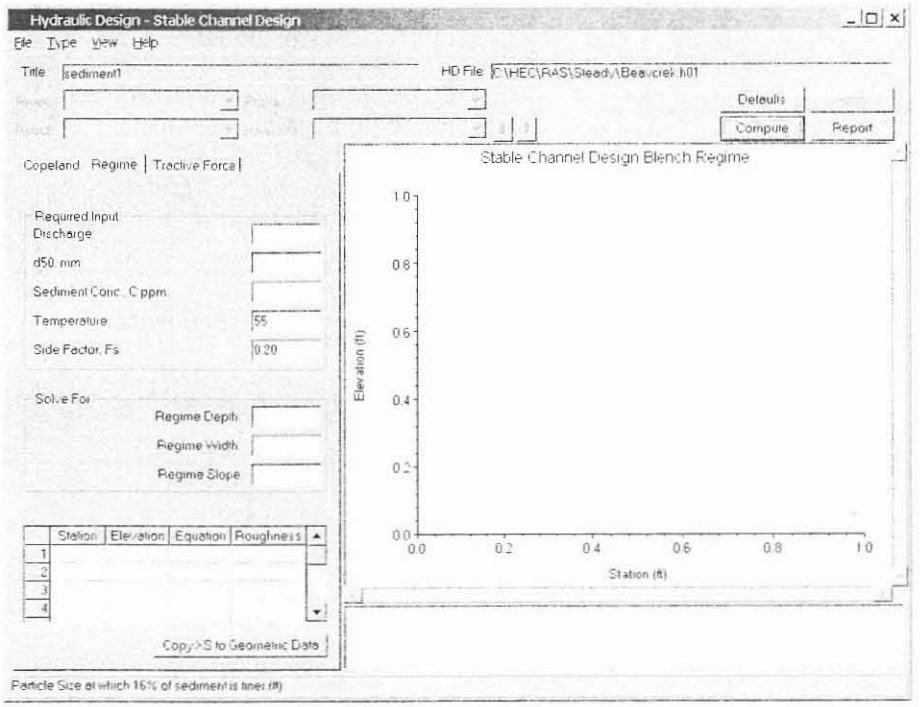


Figure 15.10 Regime Method

Enter in all required input which are:

Discharge: Channel forming discharge (cfs or m^3/s).

d50: Median particle size (mm).

Sediment Conc, C ppm: The bed material sediment concentration, in ppm.

Temperature: A representative temperature of the water. Default is 55 degrees F or 10 degrees C.

Side Factor, Fs: The side factor as defined by Blench. Blench suggests 0.1 for friable banks, 0.2 for silty, clayey, or loamy banks, or 0.3 for tough clay banks. Default value is 0.2.

Once these values are entered, the compute button becomes activated and the stable channel regime values for depth, width, and slope will be solved for and entered into the appropriate fields. In addition, the plot window will display the resulting cross section.

The displayed cross section can be added to the existing geometry file by clicking on "Copy XS to Geometric Data."

Tractive Force Method

To use the Tractive Force Method, select the tab named "Tractive Force." The window shown in figure 15.11 becomes active.

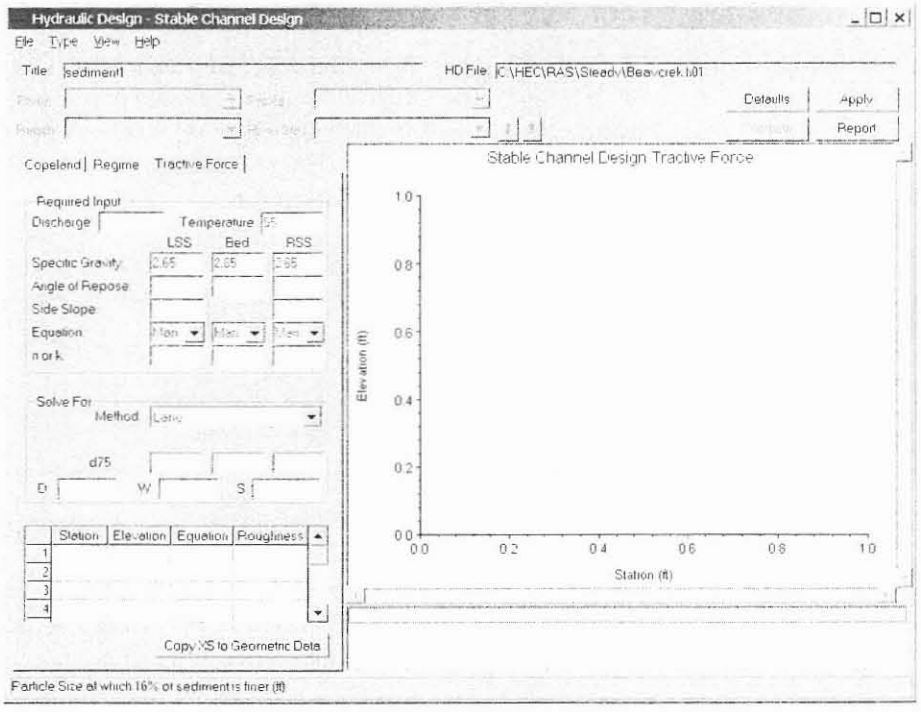


Figure 15.11 Tractive Force Method

Enter in all required input which are:

Discharge: Design discharge (cfs or m^3/s).

Temperature: Temperature of the water. Default is 55 degrees F or 10 degrees C.

Specific Gravity: Specific gravity of the sediments for the left side slope, bed, and right side slope.

Angle of Repose: The angle of repose of the sediment for the left side slope, bed, and right side slope. See figure 12-9 in the HEC-RAS Hydraulic Reference Manual for suggested values.

Side Slope: Left side slope and right side slope (1Vertical : __Horizontal).

Equation: the roughness equation for the left side slope, bed, and right side slope. Mannings and Strickler are available for use.

n or k: If Mannings is selected, enter a Mannings "n" value. If Strickler is selected, enter a "k" value for the left side slope, bed, and right side slope (ft or m for k values).

Method: Solve for critical shear using either Lane, Shields, or by entering in your own critical mobility parameter.

The remaining values are the dependant variables. Only two can be solved for at a time. The other two must be entered by the user. The three fields for particle diameter (left side slope, bed, right side slope) are considered one variable such that any one of the remaining variables plus any or all of the particle diameters can be solved for.

d50/d75: The particle diameter in which 50%/75% of the sediment is smaller, by weight. d_{50} is used for Shields and user-entered. d_{75} is used for Lane (mm).

D: The depth of the stable cross section (ft or m).

B: The bottom width of the stable cross section (ft or m).

S: The slope of the energy grade line at the stable cross section.

Once the required values plus two of the dependent variables are entered, the compute button becomes activated and the stable channel values for the remaining dependent variables will be solved for and entered into the appropriate fields. In addition, the plot window will display the resulting cross section.

The displayed cross section can be added to the existing geometry file by clicking on "Copy XS to Geometric Data."

Sediment Transport Capacity

The sediment transport capacity computations can only be run once steady or unsteady flow computations have been run. Sediment Transport Capacity for any cross section can be computed using any of the following sediment transport functions:

- Ackers-White
- Engelund-Hansen
- Laursen
- Meyer-Peter Müller
- Toffaleti
- Yang

To access the sediment transport capacity window, click on Type…Sediment Transport Capacity in the Hydraulic Design Window. The following window will become active:

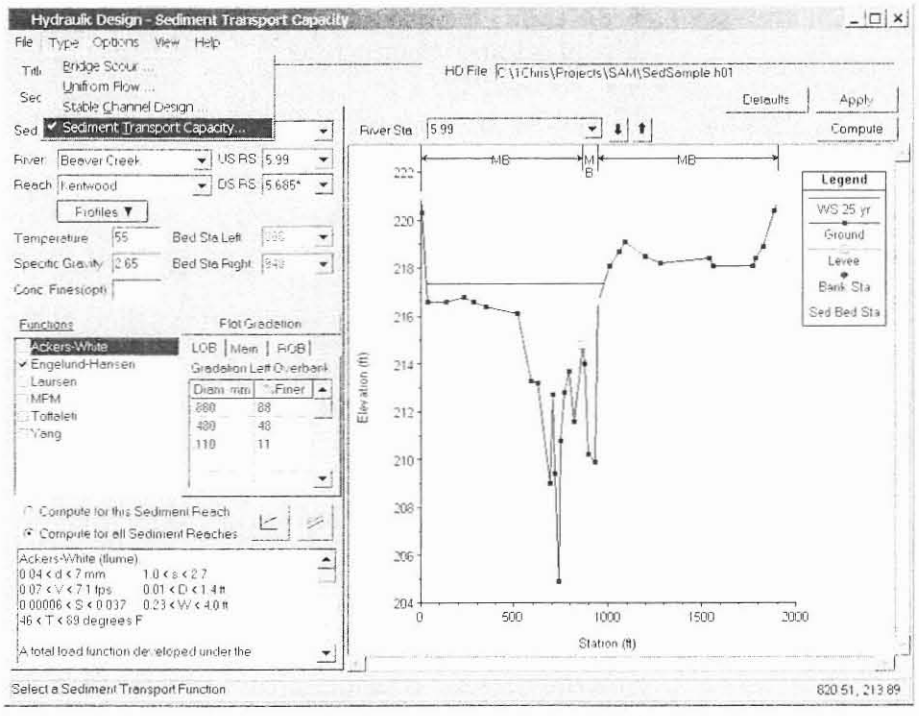


Figure 15.12 Sediment Transport Capacity Window

To perform sediment transport capacity computations, the user must define one or more sediment reaches. A sediment reach indicates for which cross sections transport rates will be computed and contains information necessary to fulfill the computations. Sediment reaches can vary spatially within the geometry, can have different input parameters such as temperature, specific gravity, and gradation, or can simply use different sediment transport

functions. A sediment reach cannot span more then one river reach, however there can be multiple sediment reaches within one river reach. **Sediment reaches cannot have overlapping cross sections.**

When the sediment transport capacity window is opened, if there are not any previously defined sediment reaches defined for the current hd file, the user will be automatically prompted to name a new sediment reach. To create a new reach otherwise, click on File…New Sediment Reach. The user also has the option of copying, deleting, and renaming existing sediment reaches under the File menu option. The name selected for the new sediment reach will appear in the Sed. Reach dropdown box along with all other existing sediment reaches for the particular hydraulic design file.

Once a new sediment reach has been named, the user must define its spatial constraints by selecting the river, reach, and the bounding upstream and downstream river stations. Next, one of the existing profiles must be selected.

Sed.Reach: Indicates which sediment reach is active. This dropdown box lists all existing sediment reaches for the current hydraulic design file.

River: The river where the current sediment reach is located.

Reach: The reach where the current sediment reach is located.

US RS: The upstream bounding river station of the current sediment reach.

DS RS: The downstream bounding river station of the current sediment reach.

Profiles: The profile to be used in the sediment transport computations for the current sediment reach.

River Sta: The river station currently displayed on the plot.

Temperature: Temperature of the water. Default is 55 degrees F or 10 degrees C.

Specific Gravity: Specific gravity of the moveable sediments. Default is 2.65.

Bed Sta Left/Right: The cross section stations that separate the left overbank from the main channel from the right overbank for sediment transport capacity computations. Defaults are the main bank stations. These values can be changed for every cross section within the sediment reach. The selected stations appear on the cross section plot as yellow nodes, and are bracketed by "MB" (mobile bed) location arrows on the top of the plot.
Conc. of Fines (opt),: The concentration of fine sediments (wash load) in the current sediment reach. This is an optional value and is used to adjust the transport rate based on Colby's (Colby, 1964) findings regarding the effects

of fine sediment and temperature on kinematic viscosity, and consequently particle fall velocity. Values are given in parts sediment per one million parts water, by weight.

Functions: The user can select one or more sediment transport functions from this list box. By clicking the checkbox, a check will appear and RAS will compute for that function. When clicking on the name of the function, a brief description of the function and its applicability will appear in the text box below.

Gradation: This is entered for the left overbank (LOB), main channel (Main) and right overbank (ROB) as defined by the left and right bed stations. The user can enter nothing or up to 50 particle size/percent finer relationships. By right-clicking on one of the tabs, the grid can be expanded for easier viewing. Right-click again to return the grid to its compact display. Typically 5 to 10 gradation points are enough to represent a typical gradation curve. The particle diameter is entered in mm under the column header Diam, mm, and the percent of the representative sediment that is finer than that particle diameter is entered under the column header %Finer. RAS then takes this gradation input to determine the fraction of the sediment that is in each standard grade size class. If a zero percent value and/or a 100% value are not entered by the user, the program will assign zero percent to the next lowest grade class and 100% to the next highest grade class. See the hydraulic reference manual for more detail.

Plot Gradation: This button gives the user a graphical representation of the sediment gradation.

The user has the option to compute sediment transport capacity rates for the currently selected sediment reach (**Compute for this Sediment Reach**) or for all existing sediment reaches (**Compute for all Sediment Reaches**) within the currently opened hydraulic design file.

A text box is provided for brief descriptions of selected transport functions. In addition to a summary of the selected function, the range of input parameters, from both field and laboratory measurements, used in the development of the respective function is also provided. Where available, these ranges are taken from those found in the SAM package user's manual (Waterways Experiment Station, 1998) and are based on the developer's stated ranges when presented in their original papers. The ranges provided for Engelund and Hansen are taken from the database (Guy, et al, 1966) primarily used in that function's development.

The following variables are used in the summaries:

- d, overall particle diameter
- dm, median particle diameter
- s, sediment specific gravity
- V, average channel velocity
- D, channel depth

- S, energy gradient
- W, channel width
- T, water temperature

Defaults: The Defaults button will restore all input boxes for the currently selected sediment reach to the default values.

Apply: The Apply button will be enabled any time new input has been added which has not been stored into memory. By clicking on the Apply button, all input for the current sediment reach will be stored to memory.

Compute: The compute button will be enabled once all required input is entered. Pressing the compute button initiates the computations for sediment transport capacity.

Options Menu: The Options Menu drop down list is on the top of the Sediment Transport Capacity form and includes:

Fall Velocity: This option allows the user to select the method of fall velocity computation. If "Default" is selected, the method used in the research and development of the respective function is chosen. Otherwise, any functions used in the computations will use the selected fall velocity method. The three fall velocity methods available are: Toffaleti, Van Rijn, and Rubey.

Depth/Width: This allows the user to select which depth and width parameters to use in the solution of the transport functions. If "Default" is selected, the program will use the depth/width combination used in the research of the selected functions(s). If any of the other depth/width combinations is used, all selected functions will be solved using those specific parameters.

Eff. Depth/Eff. Width: Used in HEC 6, this is the effective depth and effective width. Effective Depth is a weighted average depth and the effective width is calculated from the effective depth to preserve $aD^{2/3}$ for the cross section:

$$EFD = \frac{\sum_{i=1}^{n} D_{avg} a_i D_{avg}^{\frac{2}{3}}}{\sum_{i=1}^{n} a_i D_{avg}^{\frac{2}{3}}} \qquad EFW = \frac{\sum_{i=1}^{n} a_i D_{avg}^{\frac{2}{3}}}{EFD^{\frac{5}{3}}}$$

Hyd. Depth/Top Width: The hydraulic depth is the area of the cross section divided by the top width.

Hyd. Radius/Top Width: The hydraulic radius is the Area divided by the wetted perimeter. Is equivalent to hydraulic depth for relatively wide, shallow streams.

Hiding Factor for Ackers-White: An optional "hiding factor" adjustment is

available for the Ackers-White function only. The user can choose whether or not to use this feature. The default is "No."

Compute for Small Grains Outside Applicable Range: By default, RAS will perform calculations for grain sizes which are smaller than the applicable range of a given transport function. By selecting "No", the user can override this and have RAS compute for only the grain sizes within the applicability range of each sediment transport function, as defined in Table 12.7 in the Reference Manual.

Sediment Rating Curve Plot/Table: This button displays a plot of the sediment transport capacity rates for a selected cross section within a sediment reach. It is only enabled once computations for that reach have been performed. Display options can be selected from the dropdown buttons. Figure 15.13 shows a sediment rating curve plot. In addition to viewing the plots, the table tab can be clicked to view in tabular form.

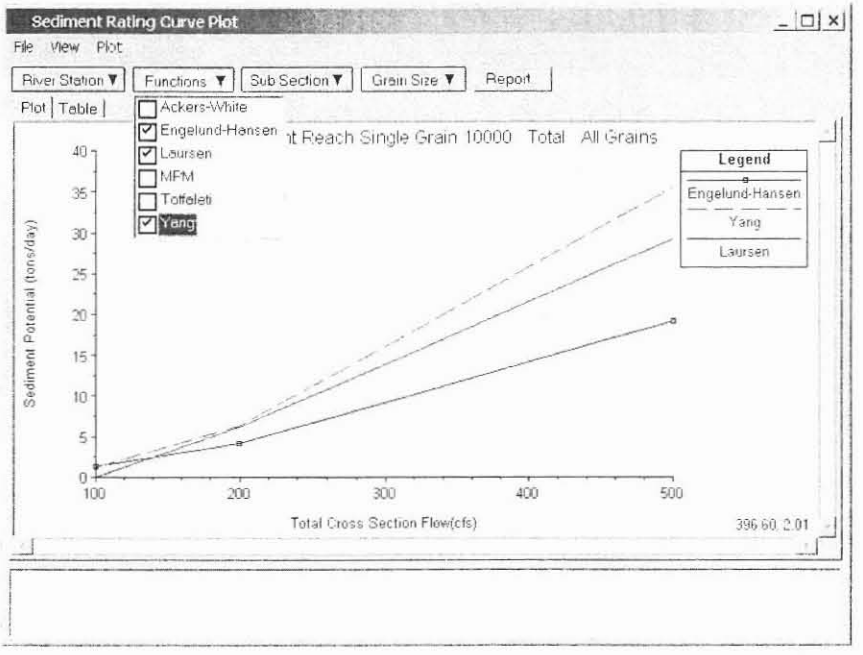


Figure 15.13 Sediment Transport Capacity Rating Curve

Sediment Transport Profile Plot/Table: This button displays a plot of the sediment transport capacity rates along a selected sediment reach. It is only enabled once computations for that reach have been performed. Display options can be selected from the dropdown buttons. Figure 15.14 shows the sediment transport profile plot. In addition to viewing the plots, the table tab can be clicked to view in tabular form.

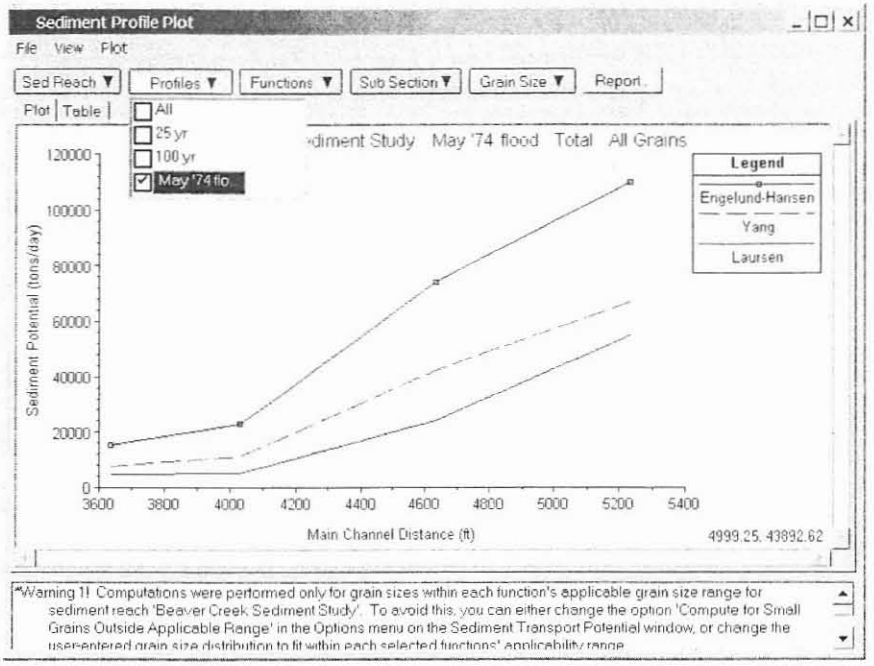


Figure 15.14 Sediment Transport Capacity Plot

Both plot windows have a list box at the bottom with warning messages. These warnings are meant to make the user aware of how sediment transport rates are being computed. If the user selects the option to compute sediment transport rates for all grade sizes within the user-specified range, a warning stating this will be shown. If the user selects the option to compute sediment transport rates for only those grade sizes within the respective function's applicability range, then a warning a different warning message will appear.

The "Compute for Small Grains Outside Applicability Range" option is located in the menu item "Options" on the Hydraulic Design window for sediment transport capacity.

Report: The Report button is located in the plot window and generates a report summarizing the input and output data. The output data is displayed as per the selections made in the dropdown buttons. Because the amount of output has the potential for being quite large, the report that is generated can likewise be very large. Figure 15.15 shows an example of the sediment transport capacity report. As with other report windows found in HEC-RAS, the user has the ability to send this report to the clipboard, print it, or save it as a text file.

Hydraulic Design Data

Sediment Reach Beaver Creek Sediment Study
River: Beaver Creek, Reach: Kentwood
RS: 5.99 to 5.685*
Sediment Transport Functions: Ackers-White, Engelund-Hansen, Laursen, MPM, Toffaleti, Yang
Temperature: 55
Specific Gravity of Sediment: 2.65
Concentration of Fine Sediment: 100
Fall Velocity Method: Default
Depth/Width Type: Hydraulic Radius/TopWidth
Hiding Factor NOT USED for Ackers-White.

Gradation	Left Overbank		Main Channel		Right Overbank	
	Diameter	% Finer	Diameter	% Finer	Diameter	% Finer
	1.00	100	1.00	100	1.00	100
	.880	88	.880	88	.880	88
	.480	48	.480	48	.480	48
	.110	11	.110	11	.110	11
	.00200	0	.00200	0	.00200	0
d90	.899		.899		.899	
d84	.828		.828		.828	
d50	.495		.495		.495	

Bed Material Fraction by Standard Grade Size

Class	dm (mm)	Left	Main	Right
1	.003	0.019	0.019	0.019
2	.006	0.019	0.019	0.019
3	.011	0.019	0.019	0.019
4	.023	0.019	0.019	0.019
5	.045	0.018	0.018	0.018
6	.068	0.048	0.048	0.048

Clipboard | Print... | File... | Close

Figure 15.15 Sediment Transport Capacity Report

CHAPTER 16

Advanced Features for Unsteady Flow Routing

HEC-RAS has several advanced features that can be used when modeling complex unsteady flow situations. These features include mixed flow regime capabilities (subcritical, supercritical, hydraulic jumps, and draw downs); the ability to perform a dam break analysis; levee overtopping and breaching; and hinge pool calculations for navigation dams.

Content:

- Mixed Flow Regime
- Dam Break Analysis
- Levee Breaching and Overtopping
- Pump Stations
- Navigation Dams

Mixed Flow Regime

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

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

$$\sigma\left[\frac{\partial Q}{\partial t}+\frac{\partial\left(\frac{\beta Q^2}{A}\right)}{\partial x}\right]+gA\left(\frac{\partial h}{\partial x}+S_f\right)=0 \qquad (16\text{-}1)$$

and

$$\sigma = F_T - F_r^m \quad (F_r \le F_T\ ;\ m \ge 1)$$
$$\sigma = 0 \quad (F_r > F_T)$$

where:

σ	=	LPI factor to multiply by inertial terms.
F_T	=	Froude number threshold at which factor is set to zero. This value should range from 1.0 to 2.0 (default is 1.0)
F_r	=	Froude number.
m	=	Exponent of equation, which changes shape of the curve. This exponent can range between 1 and 128 (default value is 10).
h	=	Water surface elevation
S_f	=	Friction slope
Q	=	Flow rate (discharge)
A	=	Active cross sectional area
g	=	Gravitational force

The default values for the equation are $F_T = 1.0$ and m = 10. When the Froude number is greater than the threshold value, the factor is set to zero. The user can change both the Froude number threshold and the exponent. As you increase the value of both the threshold and the exponent, you decrease stability but increase accuracy. As you decrease the value of the threshold and/or the exponent, you increase stability but decrease accuracy. To change either the threshold or the exponent, select **Mixed Flow Options** from the **Options** menu of the Unsteady Flow Analysis window. When this option is selected, the mixed flow regime options window will appear as shown in Figure 16.1.

As shown in Figure 16.1, the graphic displays what the magnitude of the LPI factor will be for a give Froude number and a given exponent m. Each curve on the graph represents an equation with a threshold of 1.0 (F_T) and a different exponent (m).

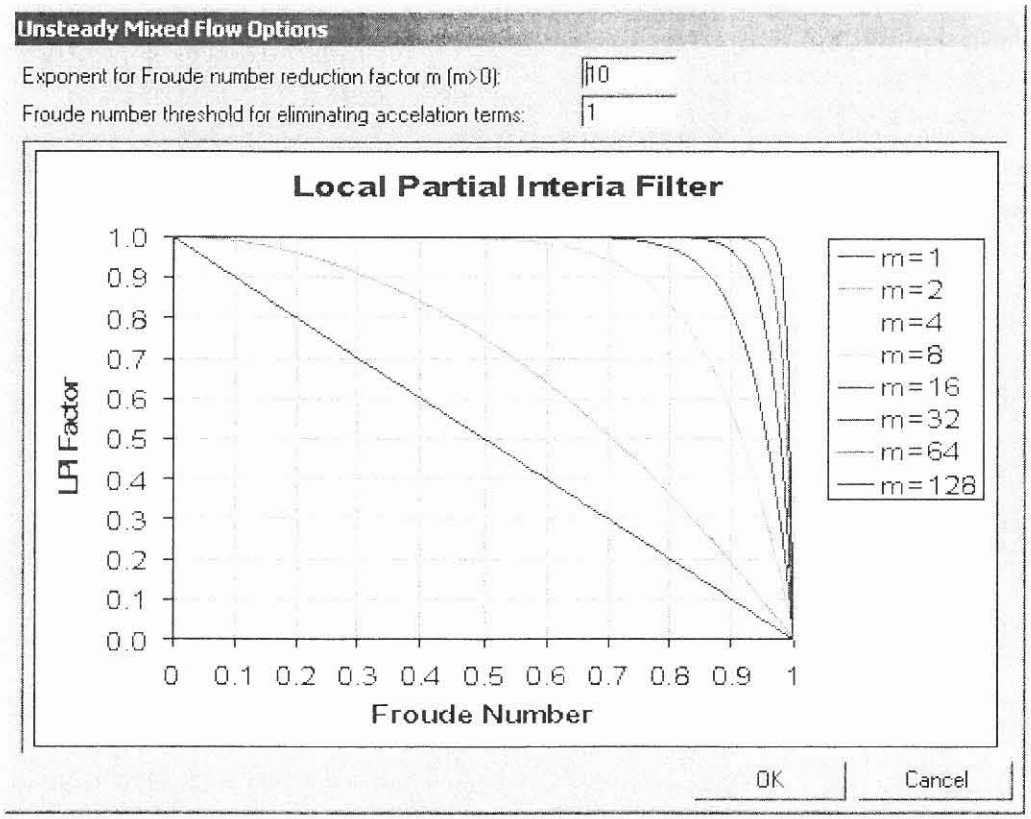


Figure 16.1 Mixed Flow Options Window

By default, the mixed flow regime option is not turned on. To turn this option on you simple check the **Mixed Flow Regime** box, which is contained within the computational settings area of the Unsteady Flow Analysis window. This window and option is shown in the Figure below.

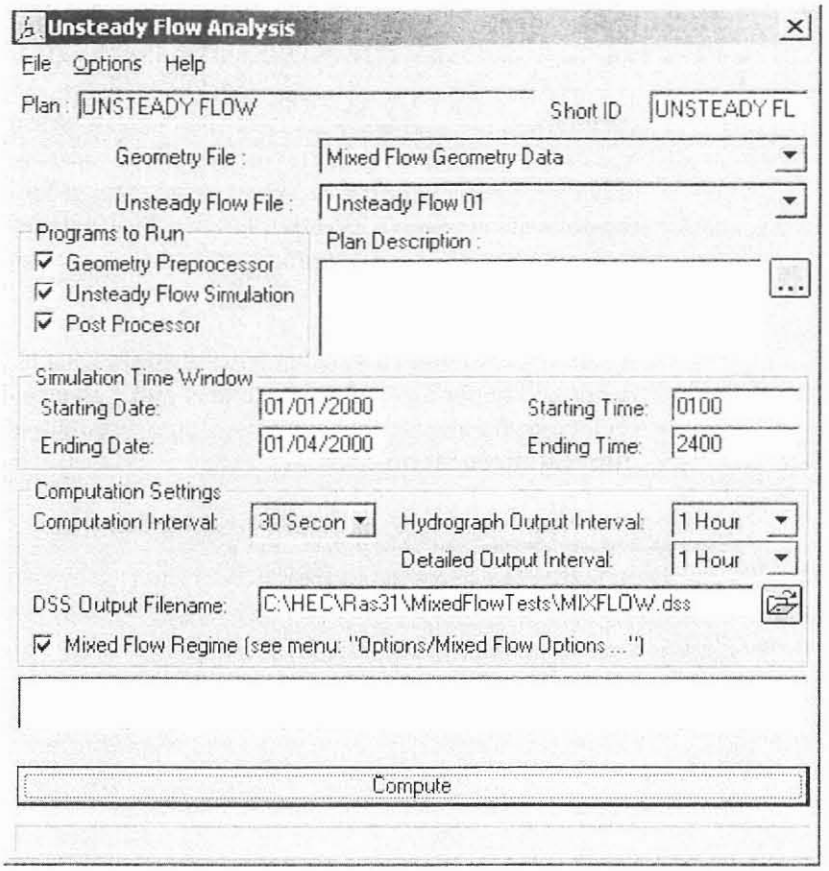


Figure 16.2 Unsteady Flow Analysis Window With Mixed Flow Regime Option Turned On.

In general, if you are working on a river system that is completely subcritical flow, you should not turn on this option. If you have a system that is mostly subcritical flow, with only a few areas that pass through critical depth, then this option can be very useful for solving the stability problems. However, there may be other options for modeling the areas that pass through critical depth. If you have some areas that are drops in the bed, and flow passes through critical depth over the drop, but is subcritical just downstream of the drop. This would be a good location to model the drop as an inline weir within HEC-RAS. By modeling the drop as an inline weir, the program is not modeling the passing through critical depth with the momentum equation, it is simply getting an upstream head water for a given flow from the weir equation. If you are modeling a system that has several areas that pass through critical depth, go supercritical, and go through hydraulic jumps, then the mixed flow methodology will be the only way to get the model to work for unsteady flow.

A profile plot of a mixed flow regime problem is shown in Figure 16.3. This example was run with the unsteady flow simulation capability within HEC-RAS, using the mixed flow regime option. The example shows a steep reach flowing supercritical, which then transitions into a mild reach. A hydraulic jump occurs on the mild reach. The mild reach then transitions back to a steep reach, such that the flow goes from subcritical to supercritical. Because of a high downstream boundary condition (for example backwater from a lake), the flow then goes from supercritical to subcritical though another hydraulic jump.

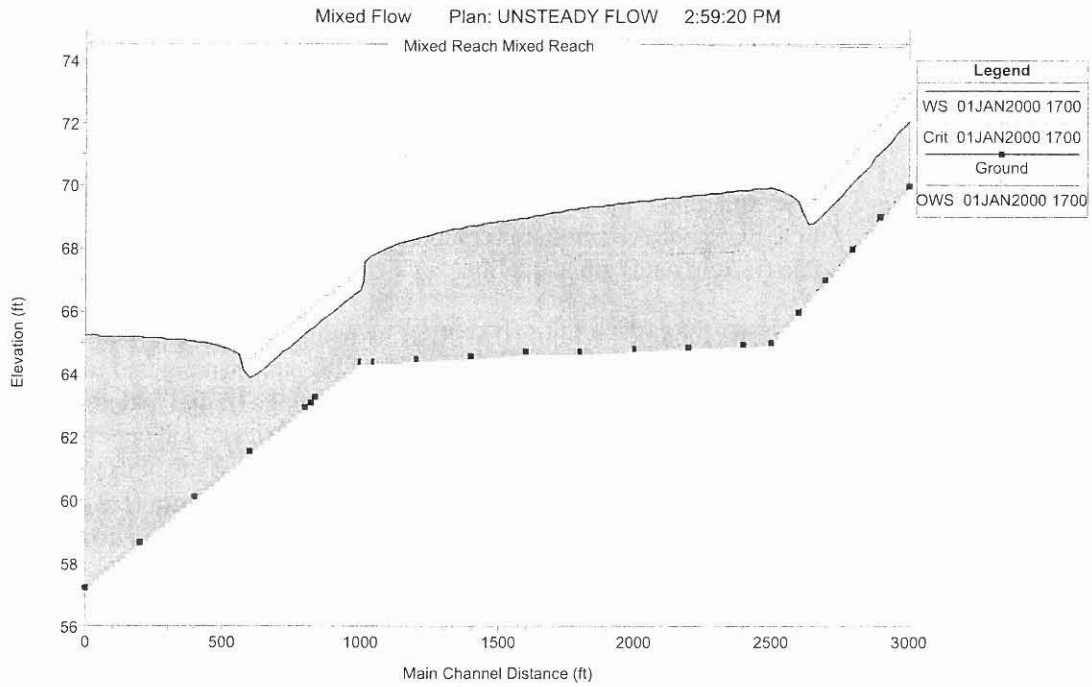


Figure 16.3 Example Mixed Flow Regime Run With Unsteady Flow Routing

Dam Break Analysis

The failure of several dams in this country (Buffalo Creek, West Virginia 1972; Teton dam, Idaho1976; Laural Run Dam and Sandy Run Dam, Pennsylvania 1977; and Kelly Barnes Dam, Georgia 1977), as well as many others, has led our nation to take a strong look at dam safety. One aspect of dam safety is to answer the question, "What will happen if the dam were to fail?" The ability to evaluate the results of a dam failure has been added into the HEC-RAS software.

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

Dams are modeled within HEC-RAS by using the Inline Structure editor. The Inline Structure editor allows the user to put in an embankment, define overflow spillways and weirs, and gated openings (radial and sluice gates). Gated openings can be controlled with a time series of gate openings or using the elevation control gate operation feature in HEC-RAS. For more information on modeling inline hydraulic structures within HEC-RAS, please review Chapter 6 of this manual (Entering and Editing Geometric Data).

An example of using the Inline Structure feature to model a dam is shown in Figure 16.4. As shown in the Figure, the user enters the embankment and overflow spillway as one piece using the Weir/Embankment editor. The embankment is shown as the gray filled in area above the ground. The overflow spillway is the rectangular notch on the upper left hand side of the embankment. The main outlet works consist of two rectangular gates, which are entered through the gate editor. The gates are shown towards the bottom of the embankment in this example.

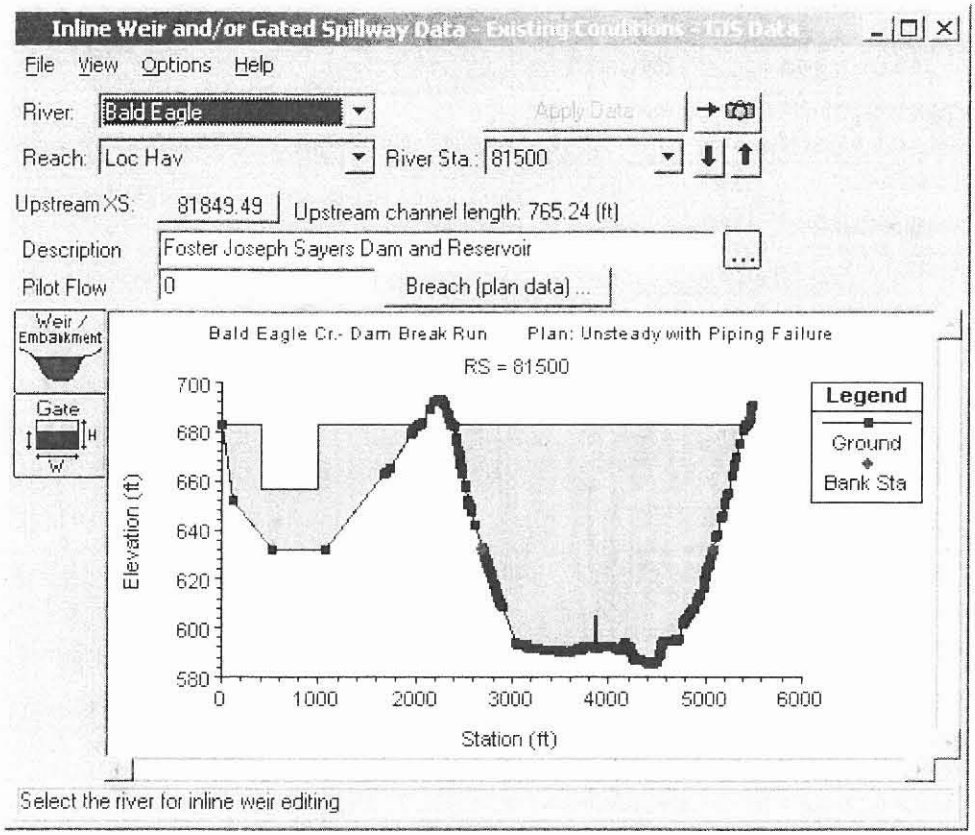


Figure 16.4 Inline Structure Editor With Example Dam Shown

Entering Dam Break Data

Entering dam breach information is accomplished by pressing the button labeled **Breach (Plan Data)**. The breach information is stored as part of the current Plan. This was done to facilitate evaluating dam and levee breaching in a real time river forecasting mode. By putting the breach information in the Plan file, the geometric pre-processor does not have to be run again, thus saving computation time during forecasting. The user can also get to the dam breach information by selecting **Dam Breach (Inline Structure)** from the Options menu of the Unsteady Flow Analysis window. Once the Breach button is pressed, the Dam Breach window will appear as shown in Figure 16.5.

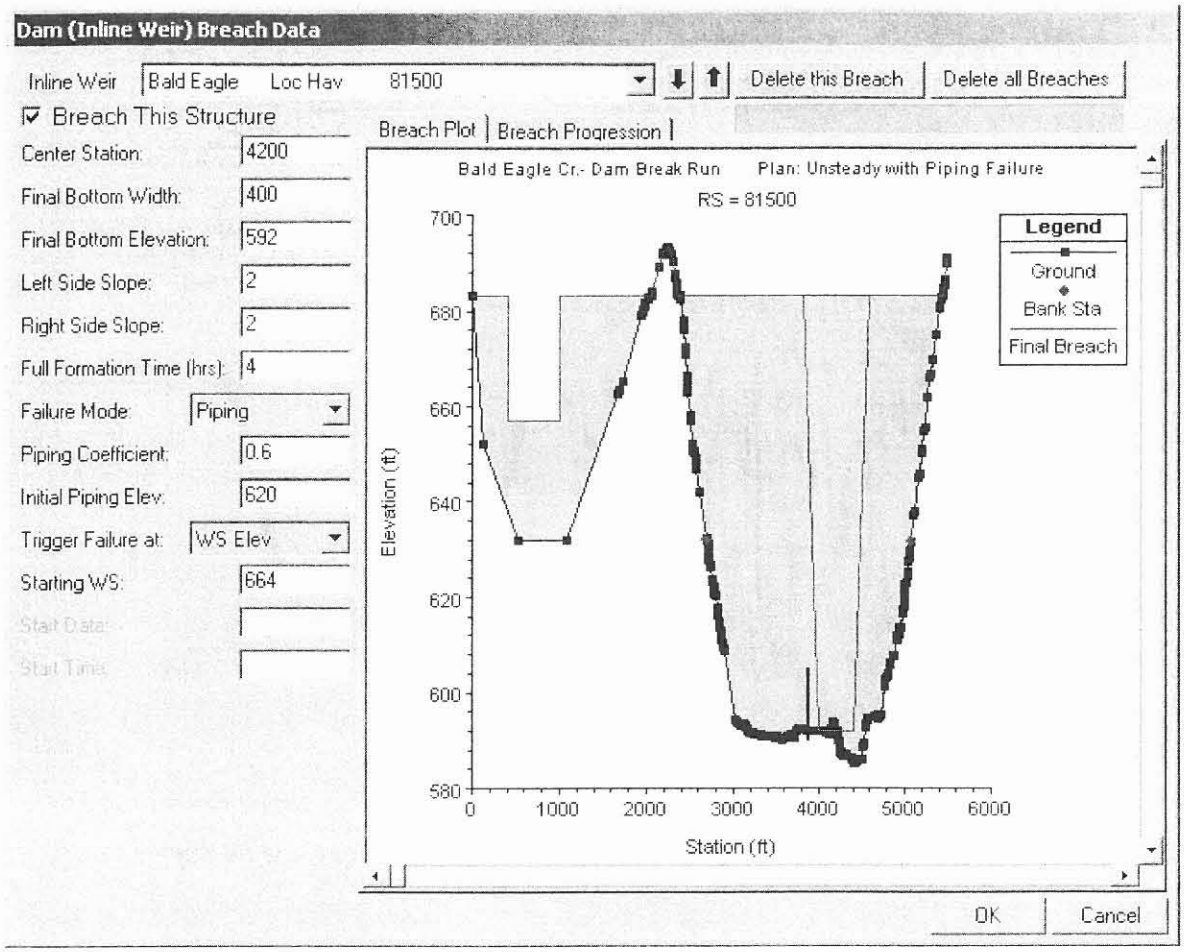


Figure 16.5 Dam Breach Data Editor With Example Dam

The data required to perform a dam breach analysis is as follows:

Inline Structure. This field is used to select the particular inline structure that you want to perform a breach analysis on. The user can enter breach data and perform a breach for more than one dam within the same model.

Delete This Breach. This button is used to clear all of the dam breach information for the currently opened inline structure.

Delete All Breaches. This button is used to delete the dam breach information for all of the inline structures in the model.

Breach This Structure. This check box is used to turn the breaching option on and off without getting rid of the breach data. This box must be checked in order for the software to actually perform the dam breach. When this box is not checked, no breaching will be performed on this structure.

Center Station. This field is used to enter the cross section stationing of the centerline of the breach. The stationing is based on the inline structure that is shown in the graphic.

Final Bottom Width. This field is used to enter the bottom width of the breach when it has reached its maximum size.

Final Bottom Elevation. This field is used to enter the bottom elevation of the breach when it has reached its maximum size.

Left Side Slope. This field is used to enter the left side slope for the trapezoid that will represent the final breach shape. If a zero is entered for both side slopes, the breach will then be rectangular. Side slopes are entered in values representing the horizontal to vertical ratio. For example, a value of 2 represents 2 feet horizontally for every 1 foot vertically.

Right Side Slope. This field is used to enter the right side slope for the trapezoid that will represent the final breach shape. If a zero is entered for both side slopes, the breach will then be rectangular. Side slopes are entered in values representing the horizontal to vertical ratio. For example, a value of 2 represents 2 feet horizontally for every 1 foot vertically.

Full Formation Time (hrs). This field is used to enter the time required for the breach to form. It represents the time from the initiation of the breach until the breach has reached its full size. The user should be very careful in selecting this time. If you are using a linear breach progression rate, you may want to limit this time to when the breach really begins to significantly erode and up to when the major portion of the breach is formed. More information on the breach full formation time will be given later in this chapter.

Failure Mode. This selection box contains two options for the failure mode of the breach, a **Piping** failure or an **Overtopping** failure. The overtopping failure mode should be selected when the water surface overtops the entire dam and erodes its way back through the embankment, or when flow going over the emergency spillway causes erosion that also works its way back through the embankment. The Piping failure mode should be selected when the dam fails due to seepage through the dam, which causes erosion, which in turn causes more flow to go through the dam, which causes even more erosion. A piping failure will grow slowly at first, but tends to pick up speed as the area of the opening begins to enlarge. At some point during the breach, the embankment above the breach will begin to sluff, at which time a large mass wasting of the embankment will occur.

Piping Coefficient. This field is only used if the Piping failure mode has been selected. The user enters an orifice coefficient into this field. The orifice equation is used to calculate the flow through the breach opening while it is acting in a piping flow manner. Once the embankment above the opening sluffs, and the water is open to the atmosphere, the program transitions to a weir equation for computing the breach flow.

Initial Piping Elev. This field is used to enter the elevation of the center of the piping failure when it first begins to occur.

Trigger Failure At. This field is used to enter the mode in which the breach initiation will be triggered. There are three options available within HEC-RAS for initiating the start of the breach: a water surface elevation (**WS Elev**), a specific instance in time (**Set Time**), and a combination of exceeding a water surface elevation for a user specified duration (**WS Elev + Duration**). With the third option (WS Elev + Duration) the user enters a threshold water surface elevation to start monitoring the location. A duration is also entered. If the water surface remains above the threshold value for the user entered duration, then the breach is initiated. Additionally the user can enter a water surface elevation labeled "**Immediate Initiation WS**." If the water surface elevation gets up to or beyond this elevation, the breach is immediately initiated.

Starting WS. This field is only used if the user has selected a trigger failure mode of water surface elevation (**WS Elev**). The user enters a water surface elevation into this field. The water surface represents the elevation at which the breach will begin to occur, once the water behind the dam has reached that elevation.

Start Date. This field is only used if the user has selected the **Set Time** option for the failure trigger mode. The user enters the date at which the breach will begin to occur. The time of the breach initiation is entered into the next field. The date should be entered in a month/day/year format (ex. 05/23/2002).

Start Time. This field is used to enter a starting time to initiate the breach. The time is entered as a military time (ex. 1800 for 6:00 p.m.).

Breach Plot. When this tab is selected, a plot of the inline structure will show up in the graphic window. The plot will show the proposed breach maximum size and location in a red color.

Breach Progression. When this tab is selected a table will appear in the graphic display window. The table is used to enter a user defined progression curve for the formation of the breach. This is an optional feature. If no curve is entered, the program automatically uses a linear breach progression rate. This means that the dimensions of the breach will grow in a linear manner during the time entered as the full breach formation time. Optionally, the user can enter a curve to represent the breach formation as it will occur during the breach development time. The curve is entered as Time Fraction vs. Breach Fraction. The Time Fraction is the decimal percentage of the full breach formation time. The breach fraction is the decimal percentage of the breach size. Both factors are entered as numbers between zero and one. An example of a user entered nonlinear breach progression rate is shown in Figure 16.6.

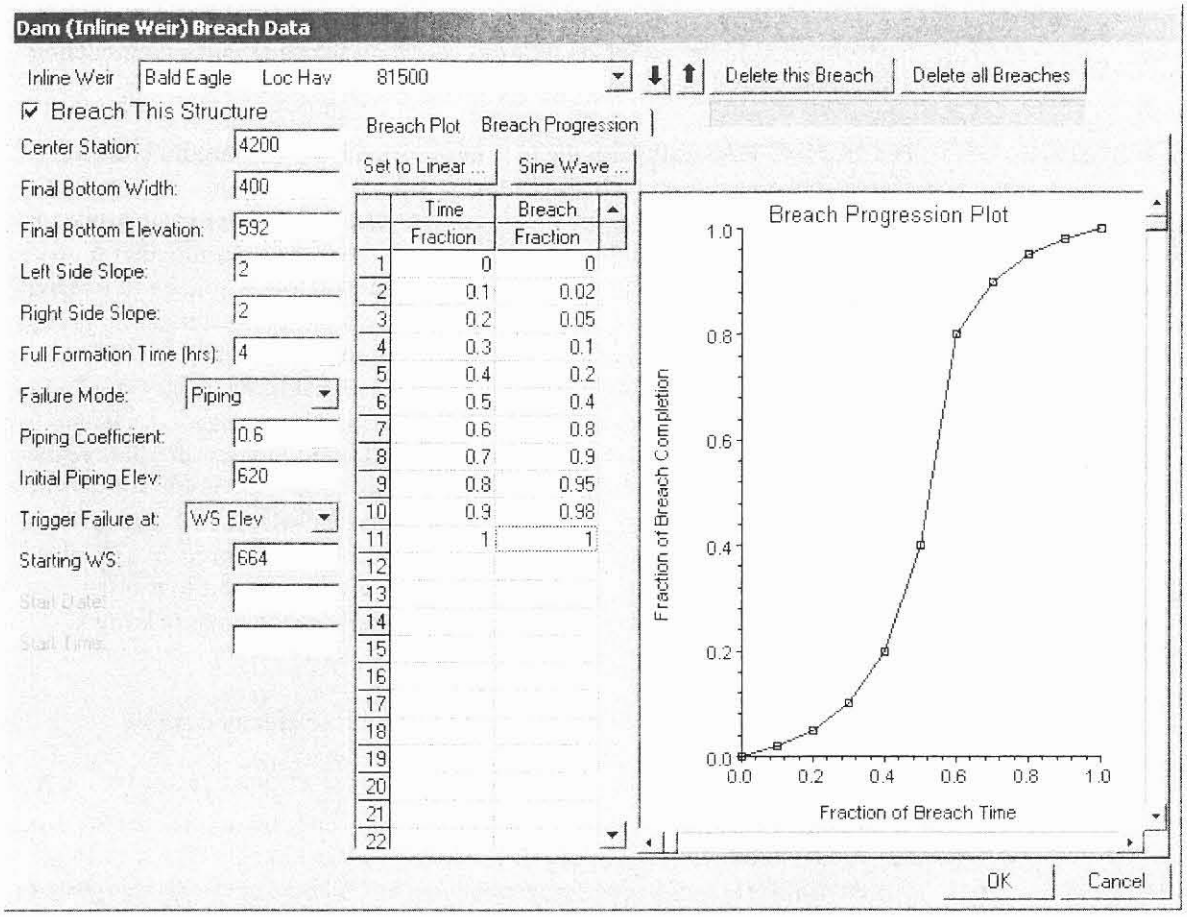


Figure 16.6 Dam Breach Editor With Nonlinear Breach Progression

Once all of the Dam Breach data are entered, press the **OK** button to have the data accepted. However, the data is not saved to the hard disk at this point, you must save the currently opened plan in order for the breach information to be save to the hard disk.

Estimating Dam Break Parameters

The key parameters that must be estimated in any dam breaching analysis are the breach formation time and the maximum size of the breach opening. Several researchers have developed regression equations to estimate breach sizes and times from historical dam breach information. Additionally a few researchers have tried to develop computer models to simulate the physical breach process. The bulk of the research in this area has been summarized in a 1998 publication entitled "Prediction of Embankment Dam Breach Parameters", by Tony L. Wahl of the U.S. Bureau of Reclamation. This publication documents the data from most of the historical dam breaches that

have occurred in the world, as well as describing the equations and modeling approaches developed for predicting the dam breach parameters.

For the HEC-RAS software, the user must estimate the maximum breach dimensions and breach formation time outside of the program. Because the breaching process is complex, it is suggested that the modeler try to come up with several estimates of the breach parameters, and then put together a matrix of potential breach sizes and times. One example would be to use two different sets of regression equations and one of the breach simulation models to estimate the breach parameters. In several studies performed at HEC we have used both the Froelich (1995) and the MacDonald\Langridge-Monopolis (1984) regression equations, as well as the BREACH model by Dr. Danny Fread (Fread, 1988). All three methods give different answers for the breach dimensions, as well as the time for the breach to form. One could simply run each of these estimates as a separate trial within HEC-RAS, or a matrix could be formed by trying all three breach times with each of the breach sizes, thus ending up with 9 different runs. Either way, it is always good to test the sensitivity of the breaching parameters, since they are the most unknown factor in this process.

Each of the breach parameter estimates will yield a different outflow hydrograph from HEC-RAS. However, once these hydrographs are routed downstream, they will tend to converge towards each other. How close they get to each other will depend on the distance they are routed, the steepness of the stream, the roughness of the river and floodplain, and the amount of floodplain storage available for attenuating the hydrograph. If the populated areas below the dam are quite a distance away (say 20 miles or more), then the resulting hydrographs from the various dam breaches may be very similar in magnitude by the time they reach the area of interest. However, if the areas of interest are closer to the dam, then the resulting breach hydrographs could produce quite a range in results. In this situation, the selection of the breach parameters is even more crucial.

HEC-RAS Output For Dam Break Analyses

Several plots and tables are available for evaluating the results of a dam break analysis within HEC-RAS. Graphics include cross section, profile, and 3 dimensional plots, all of which can be animated on a time step by time step basis in order to visualize the propagation of the flood wave. An example cross-section plot of a dam while it is breaching is shown in Figure 16.7. Additionally, the corresponding water surface profile for the same instance in time is shown in Figure 16.8. Hydrographs can be viewed at any location in which the user requested hydrograph output. Shown in Figure 16.9 is a series of hydrographs from the breach shown in the previous figures. These hydrographs represent the flow leaving the dam and then subsequent locations downstream as the flood wave moved through the river system.

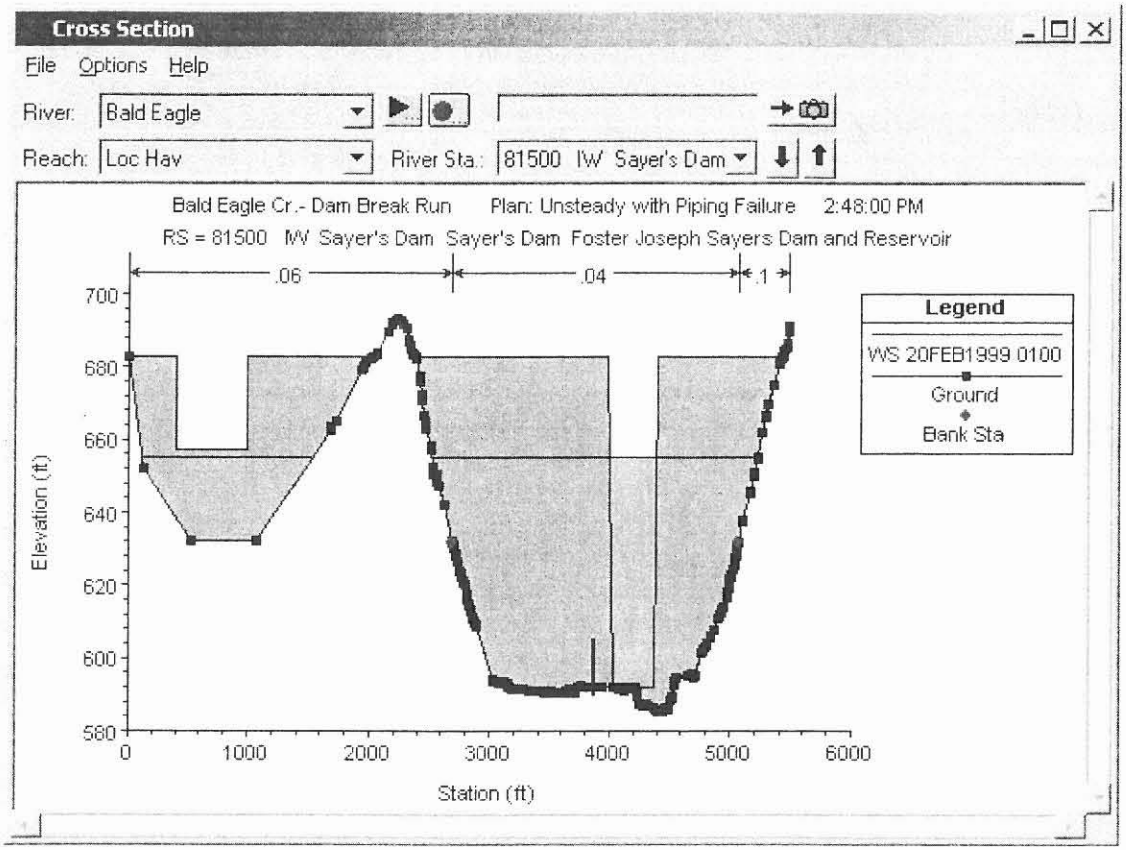


Figure 16.7 Example Plot of Dam While Breaching

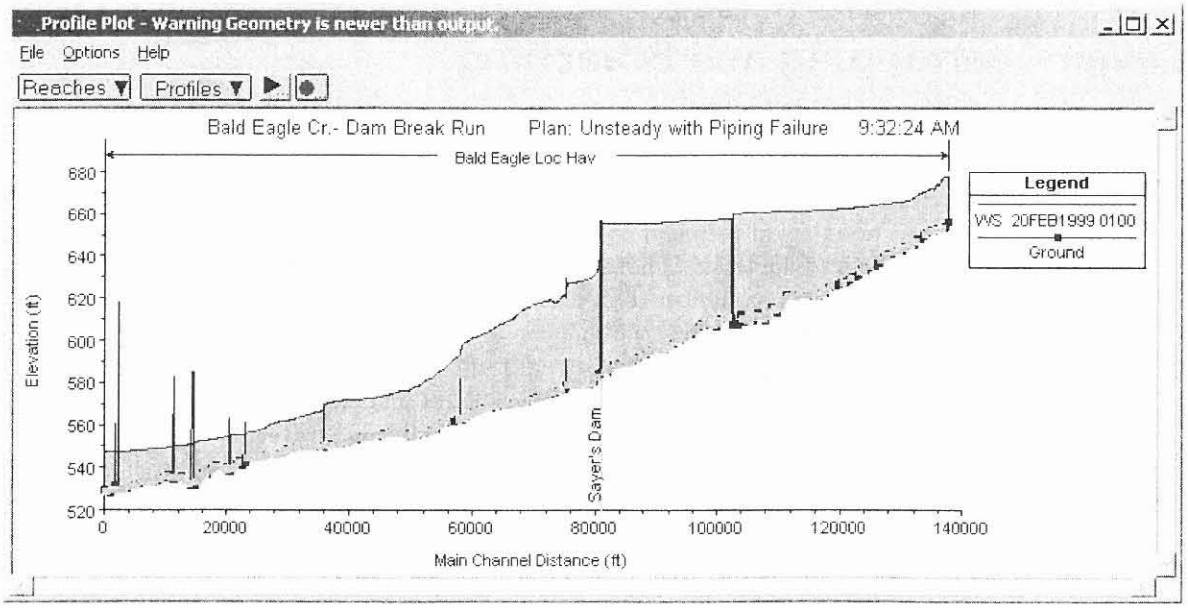


Figure 16.8 Example Profile Plot of Dam Breaching

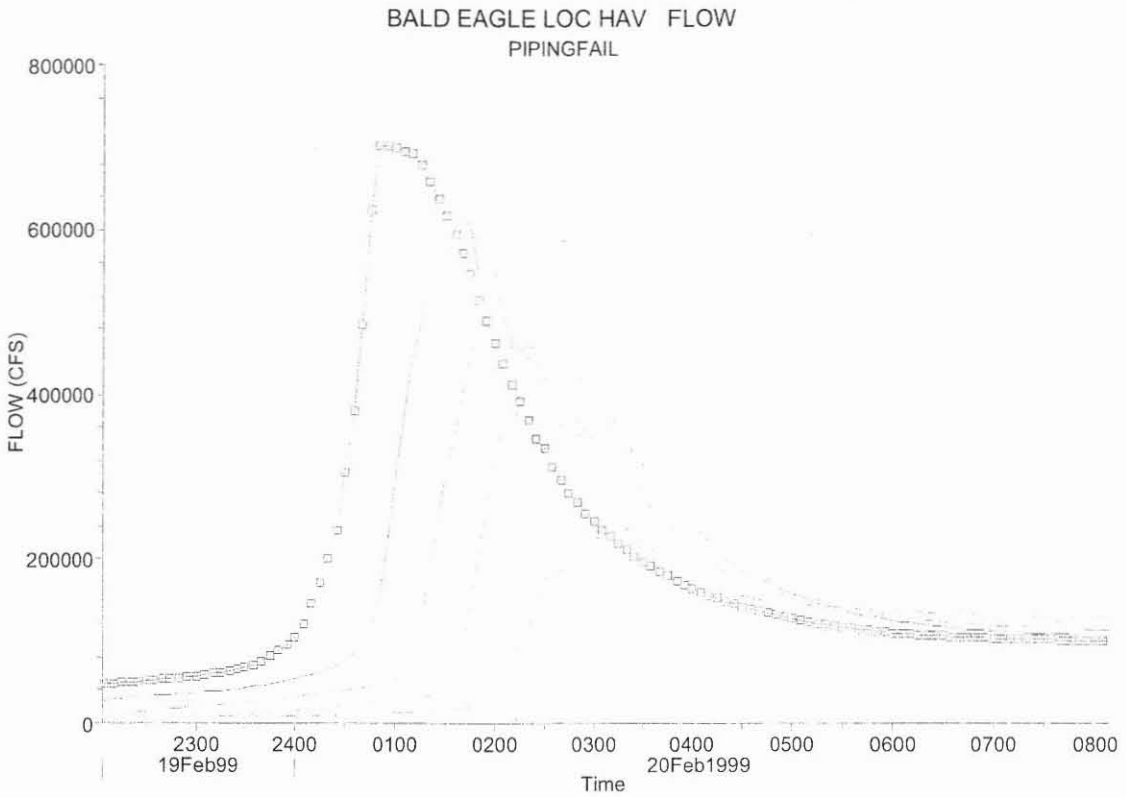


Figure 16.9 Flow Hydrographs From Dam to Downstream Locations.

Levee Overtopping and Breaching

Levee overtopping and breaching can be analyzed within HEC-RAS by modeling the levee as a lateral structure. When modeling a levee with a lateral structure, the area behind the levee should not be included in the cross section data of the main river. The cross sections should stop right at the bottom of the levee. The lateral structure (levee) can be connected to a storage area or another river reach. How you model the area behind the levee will depend upon what will happen to the water if the levee overtops or breaches. If the water going over or through the levee will pond, then a storage area would be more appropriate for modeling the area behind the levee. If the water will continue to flow in the downstream direction, and possibly join back into the main river, then it would be more appropriate to model that area as a separate river reach. Shown in Figure 16.10 is an example schematic with a levee modeled as a lateral structure connected to a storage area to represent the area behind the levee. An example cross section with a lateral structure (levee) on the right hand side is shown in Figure 16.11.

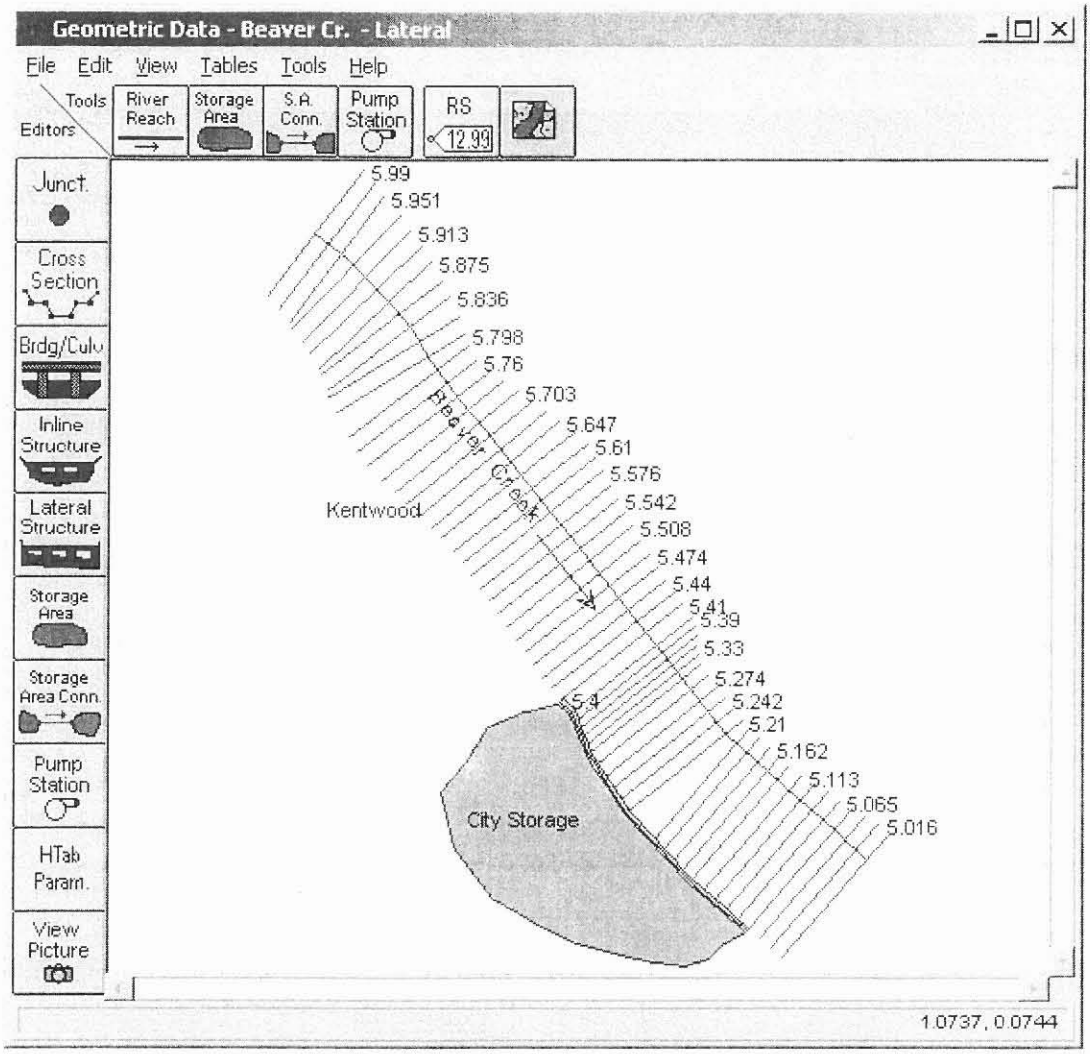


Figure 16.10 Schematic With Example Levee and Storage Area

The user defines the levee by entering a series of station and elevation points that represent the top of levee profile. This station and elevation data is then used as a weir profile for calculating the amount of water going over top of the levee. An example levee entered as a lateral structure is shown in Figure 16.12.

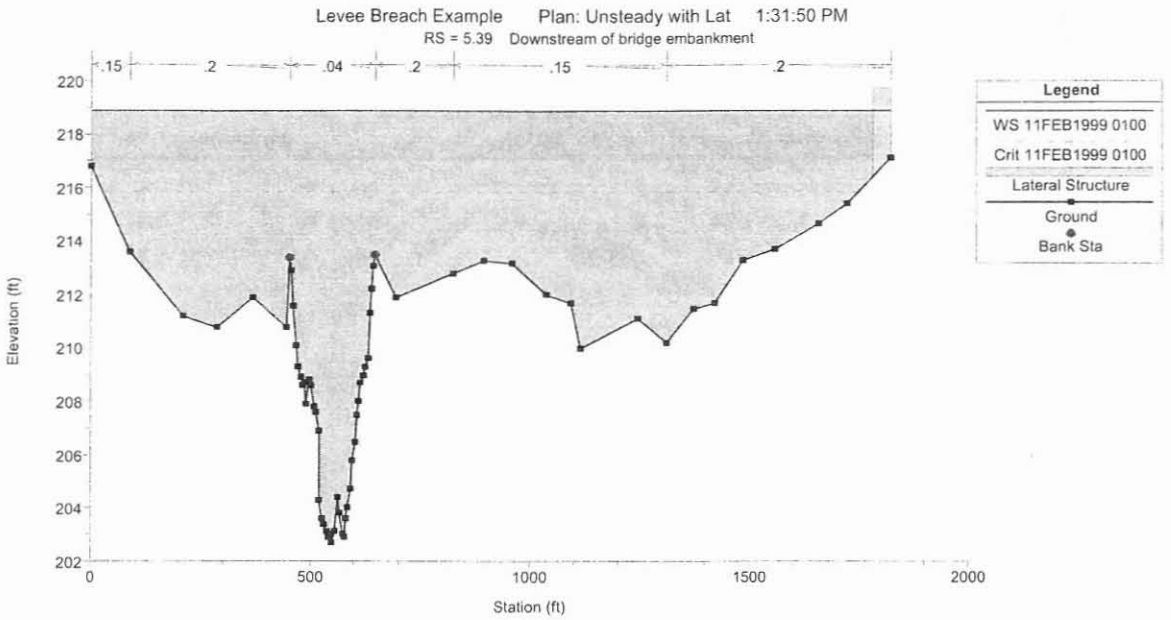


Figure 16.11 Example Cross-Section With Lateral Structure

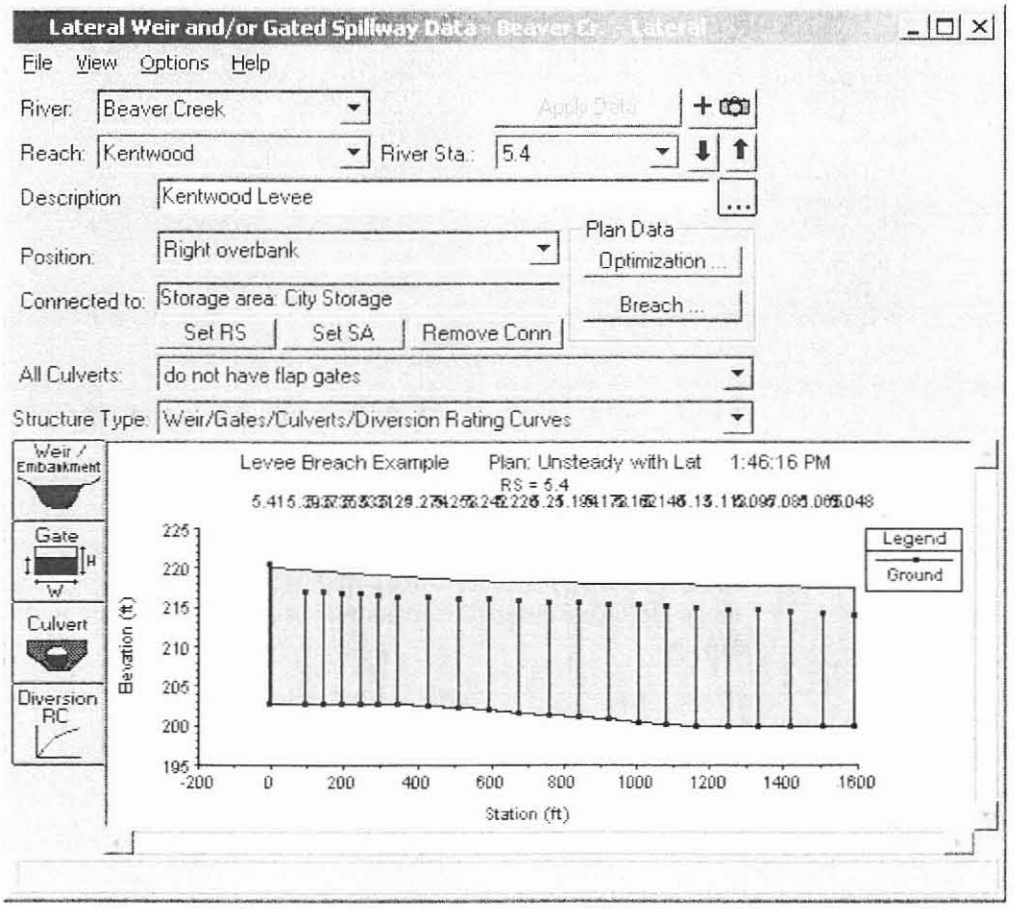


Figure 16.12 Lateral Structure Editor With Levee Modeled as a Weir

In the example shown in Figure 16.12, the levee is connected to a storage area that will be used to represent the area behind the levee. As the levee overtops and/or breaches, the storage area will fill up until it reaches the same elevation as the water in the river. After the flood passes, the water in the storage area can pass back out any breach that may have occurred.

The levee information is entered as station and elevation data in the **Lateral Weir/Embankment** editor shown on the Lateral Structure editor. The station elevation data represents the top of the levee. An example of this editor with levee data is shown in Figure 16.13.

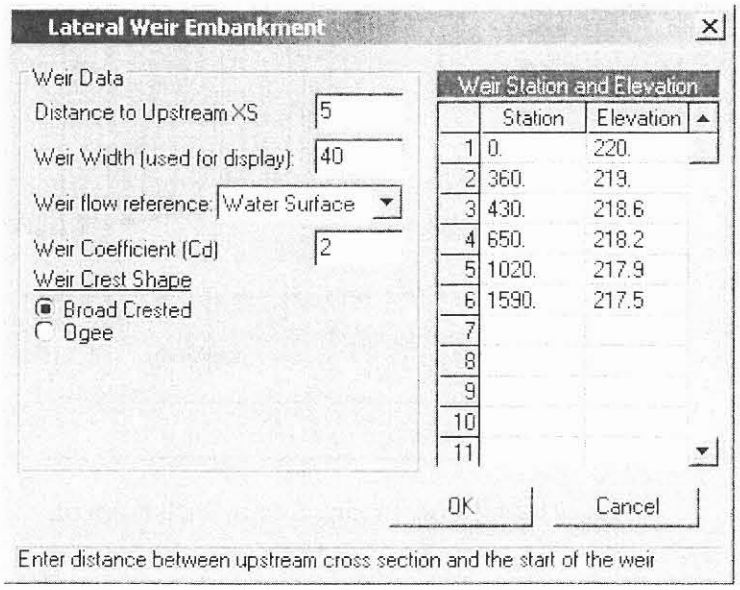


Figure 16.13 Lateral Weir/Embankment Editor With Example Levee Data

As shown in Figure 16.13, the user enters the distance that the upstream end of the levee is from the nearest upstream cross section; the width of the levee (which is only used for drawing purposes); the head reference for weir flow calculations; the lateral weir coefficient; and the station and elevation data representing the top of levee.

Once the physical levee information is entered, the user can press the **Breach** button in order to bring up the levee breach editor. An example of this editor is shown in Figure 16.14.

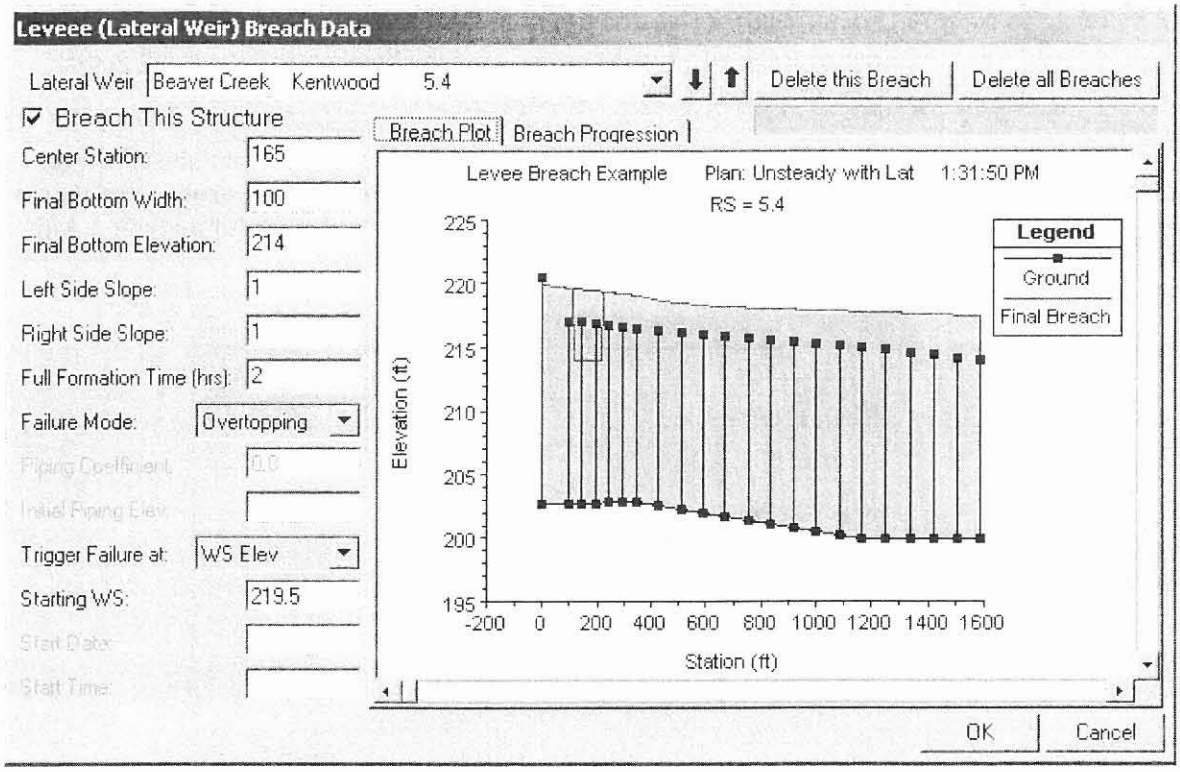


Figure 16.14 Levee Breach Editor With Example Levee and Breach

As shown in Figure 16.14, the information required to perform a levee breach is the same as performing a dam break. To get the details of each data field, please review the information found under the Dam Break section of this chapter.

After all of the data are entered and the computations are performed, the user can begin to look at output for the lateral structure (levee). Plots such as the profile plot, lateral structure hydrographs, and storage area hydrographs, can be very helpful in understanding the output for a levee overtopping and/or breach. Shown in Figure 16.15 is an example profile plot with a levee breach. Shown in Figure 16.16 is a stage and flow hydrograph plot for the lateral structure. In this plot there are three stage lines and three flow lines. The stage lines represent; the stage in the river at the upstream end of the levee (Stage HW US); the stage in the river at the downstream end of the levee (Stage HW DS); and the stage in the storage area (Stage TW). The river is always considered to be the headwater, and the storage area is the tailwater. The flow lines on the plot represent: the flow in the river at the upstream end of the levee (Flow HW US); the flow in the river at the downstream end of the levee (Flow HW DS); and the flow leaving the river over the lateral weir to the storage area (Flow Leaving).

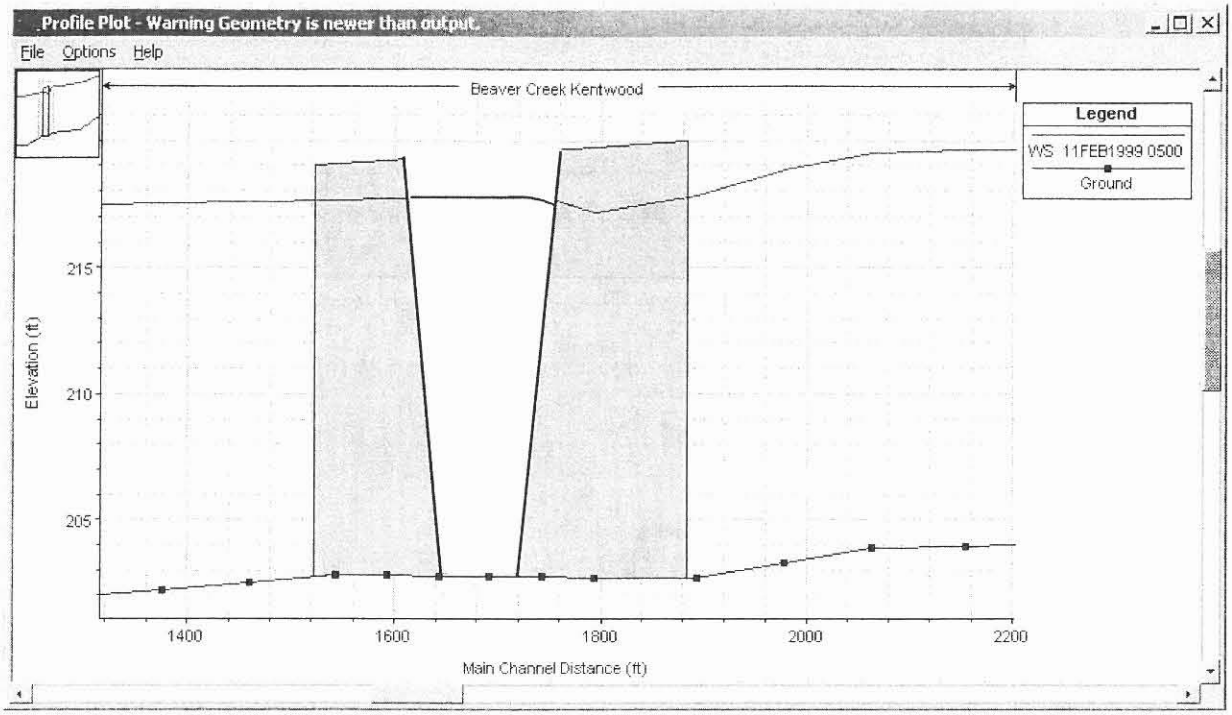


Figure 16.15 Profile Plot With Levee Breach

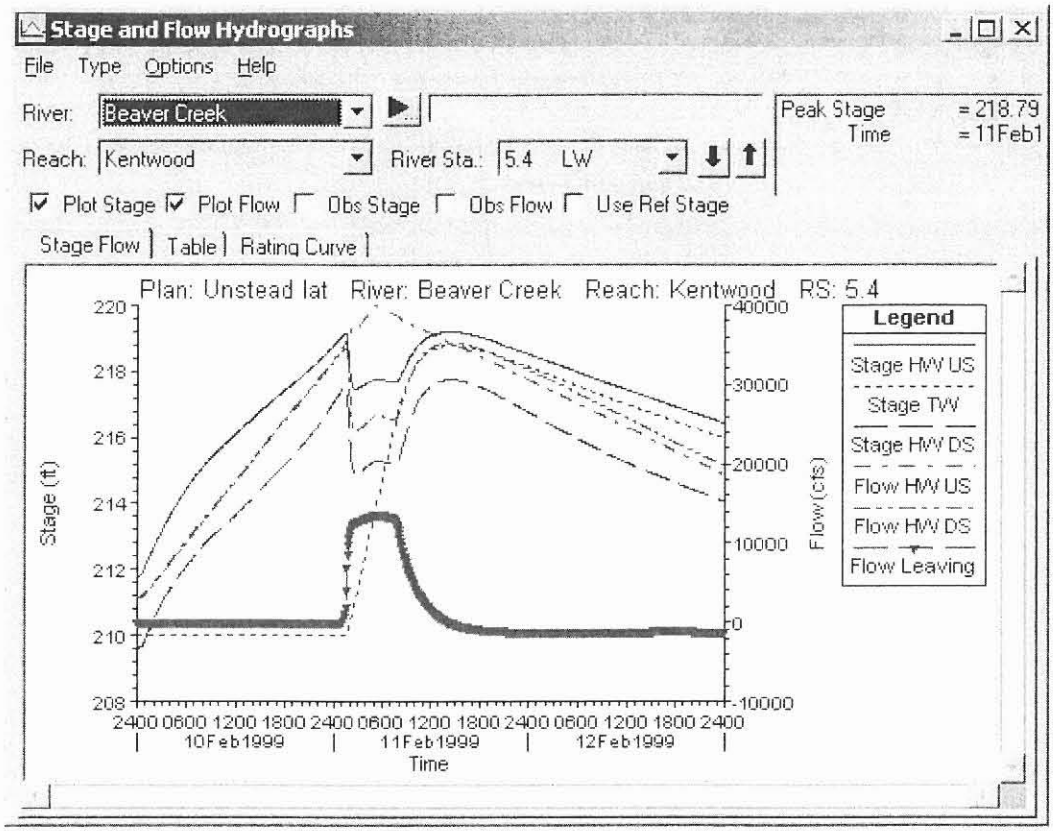


Figure 16.16 Inline Structure Stage and Flow Hydrographs

In addition to the profile plot and the lateral structure hydrographs, it is normally a good idea to plot the stage and flow hydrographs for the storage area. This allows the user to easily see the amount of flow coming into and out of the storage area, and the change in the water surface elevation. Shown in Figure 16.17 is the stage and flow hydrograph for the storage area in this example.

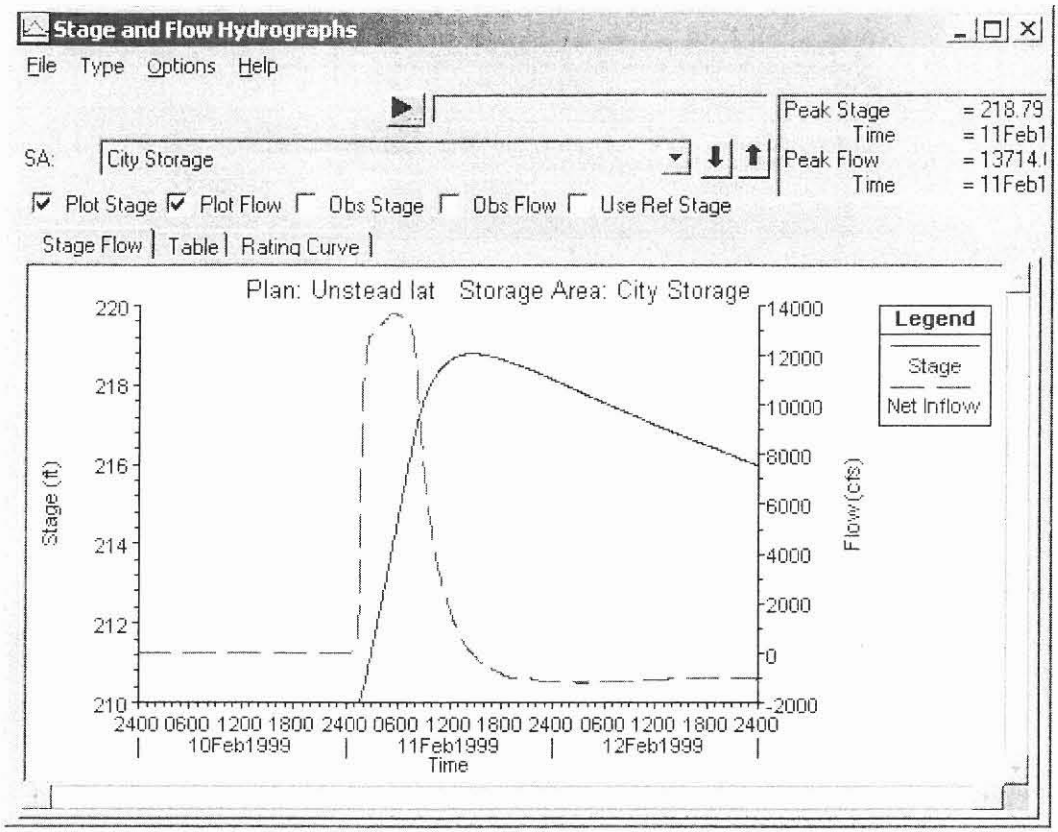
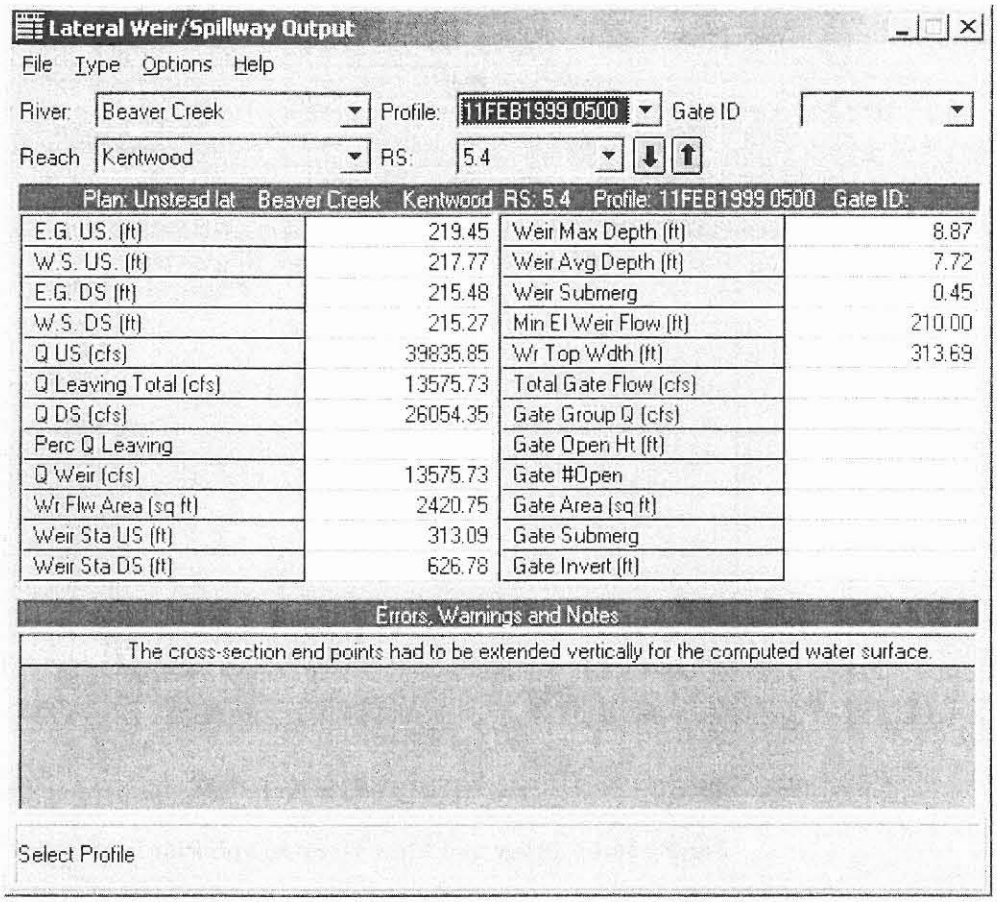


Figure 16.17 Stage and Flow Hydrograph Plot for Storage Area

Referring to Figures 16.16 and 16.17, as the levee breaches, the flow going into the storage area and the stage increase quickly, while the stage and flow in the main river drop. In addition to the graphics in HEC-RAS, tabular results are also available. Shown in Figure 16.18 is a detailed output table for the lateral structure. The user can select a specific time line for viewing the output.



Lateral Weir/Spillway Output

File Type Options Help

River: Beaver Creek Profile: 11FEB1999 0500 Gate ID

Reach Kentwood RS: 5.4

Plan: Unstead lat Beaver Creek Kentwood RS: 5.4 Profile: 11FEB1999 0500 Gate ID:

E.G. US. (ft)	219.45	Weir Max Depth (ft)	8.87
W.S. US. (ft)	217.77	Weir Avg Depth (ft)	7.72
E.G. DS (ft)	215.48	Weir Submerg	0.45
W.S. DS (ft)	215.27	Min El Weir Flow (ft)	210.00
Q US (cfs)	39835.85	Wr Top Wdth (ft)	313.69
Q Leaving Total (cfs)	13575.73	Total Gate Flow (cfs)	
Q DS (cfs)	26054.35	Gate Group Q (cfs)	
Perc Q Leaving		Gate Open Ht (ft)	
Q Weir (cfs)	13575.73	Gate #Open	
Wr Flw Area (sq ft)	2420.75	Gate Area (sq ft)	
Weir Sta US (ft)	313.09	Gate Submerg	
Weir Sta DS (ft)	626.78	Gate Invert (ft)	

Errors, Warnings and Notes

The cross-section end points had to be extended vertically for the computed water surface.

Select Profile

Figure 16.18 Detailed Tabular Output for Lateral Structure

Modeling Pump Stations

Pump stations can be connected between storage areas; a storage area and a river; and between river reaches. The user can have up to ten different pump groups, and each pump group can have up to twenty identical pumps. Each pump can have its own on and off trigger elevation. To learn how to connect a pump and how to enter pump data, please review the section on pumps in Chapter 6 of the user's manual.

Pump stations can be used for many purposes, such as pumping water stored behind a levee (interior sump) into the main river. An example schematic of an interior ponding area behind a levee is shown in Figure 16.19. Note that the pump is connected from the storage area to a river station at the downstream end of the levee.

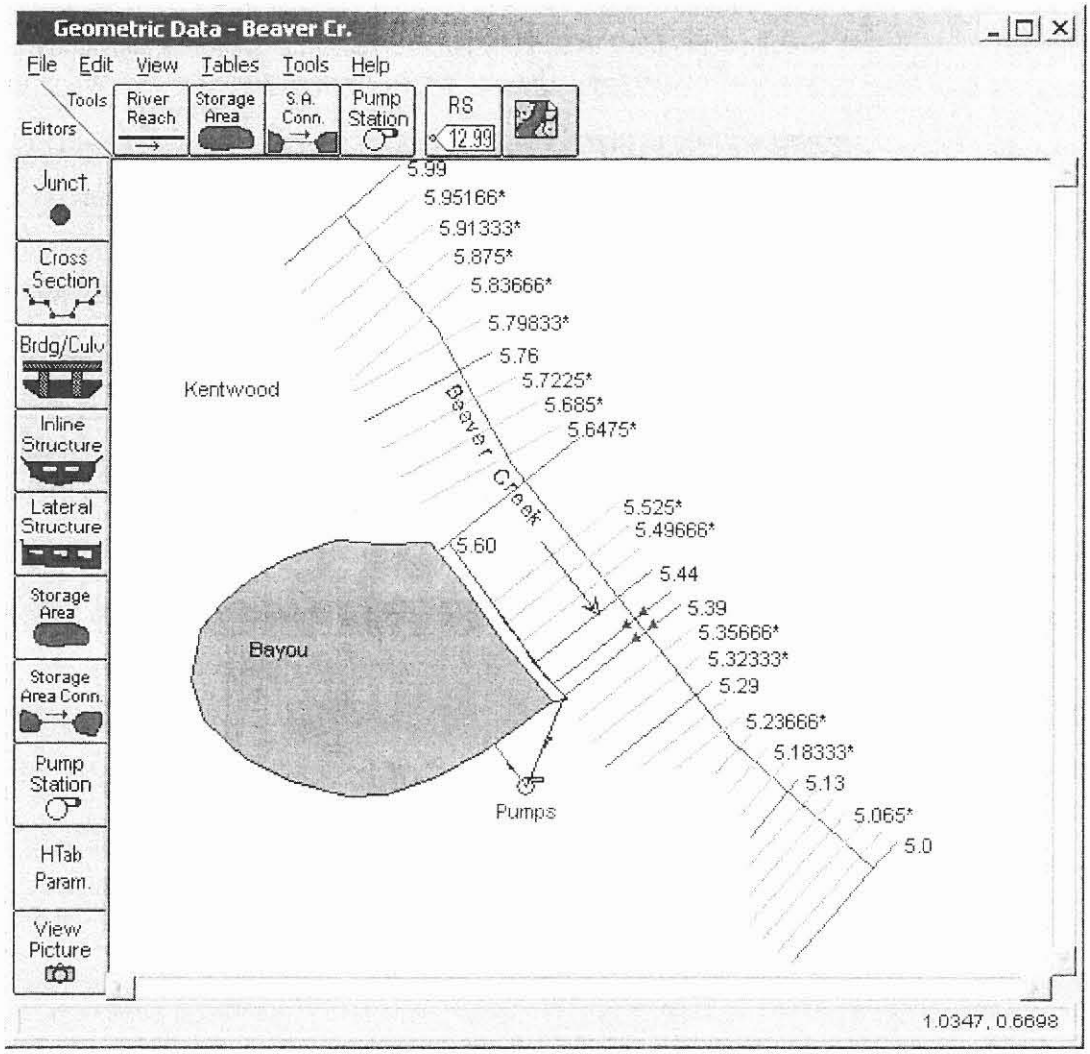


Figure 16.19 Example Pumping Station for Interior Ponding Area

In the example shown in Figure 16.19, a lateral structure was entered to represent the levee. This structure has a gravity draining culvert with a flap gate. The flap gate only allows water to drain from the storage area to the river. Additionally, a pump station is included to pump flows over the levee during a rainfall event. The pump station was drawn by selecting the **Pump Station** tool, then drawing a connection from the storage area to the cross section at river station 5.39.

In this example there is a hydrograph attached to the upstream end of the river reach, which represents the incoming flood wave to this reach. There is also a lateral inflow hydrograph attached to the storage area, which represents the local runoff collecting behind the levee. The pumps are used to pump water from the storage area, over the levee, to the river. The top of the levee is

around elevation 220 feet. Therefore, the pump station is constantly pumping to a head of 220 feet. The data for the pump station is shown in Figure 16.20.

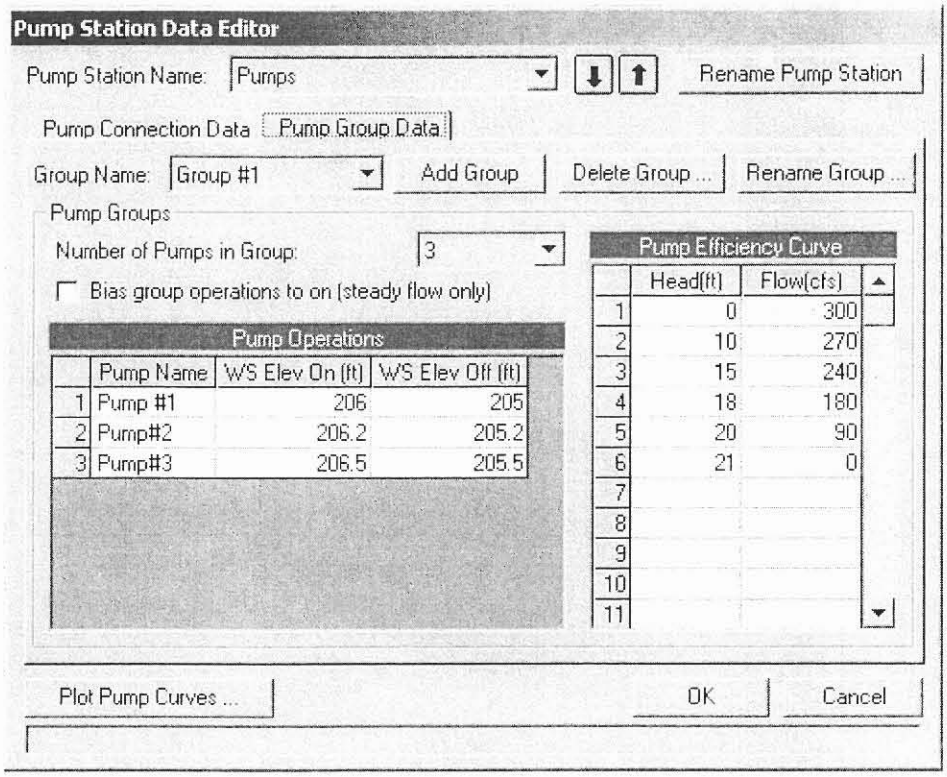


Figure 16.20 Pump Station Editor With Example Data.

As shown is Figure 6.20, there is one pump group with three identical pumps (pumps are the same size and capacity). However, each of the pumps has a different on and off trigger elevation. The pump efficiency curve is used for all three of the pumps.

After the computations are performed the user can view output for the pump station by selecting the stage and flow plotter, then selecting **Pump Stations** from the **Type** menu at the top of the window. An example stage and flow plot for the pump station is shown in Figure 16.21. As shown in the figure, the stage for the tailwater location (Stage TW) is a constant 220 ft. This is due to the fact that the pump is constantly pumping over the levee at elevation 220. The stage at the headwater location (stage HW) is the water surface elevations in the storage area. The storage area elevation starts out at an elevation of 205 ft., goes up to around 206.6, and then back down to around 205.1. The flow through the pumps was zero until an elevation of 206 was reached within the storage area. The second pump turned on when the storage area got to elevation 206.2, and the third at elevation 206.5. On the following side of the hydrograph the pumps began to turn off as the stage went down in the storage area. Shown in Figure 16.22 are the stage and net

inflow to the storage area. The net inflow represents all the inflows minus the outflows at each time step.

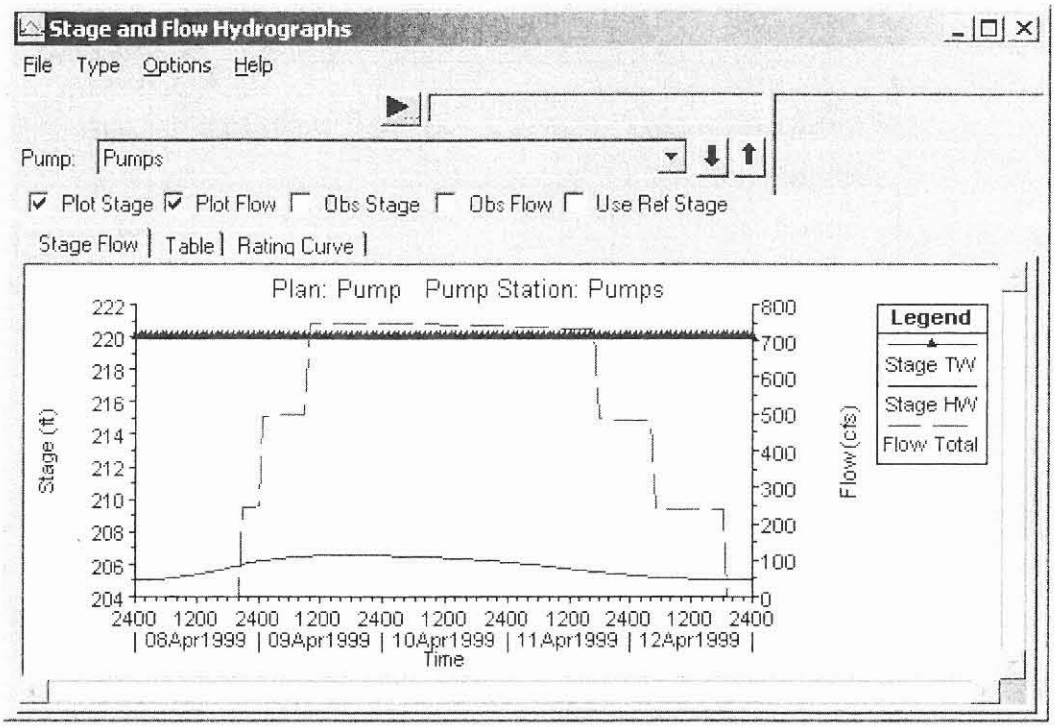


Figure 16.21 Stage and Flow Hydrographs for Pump Station

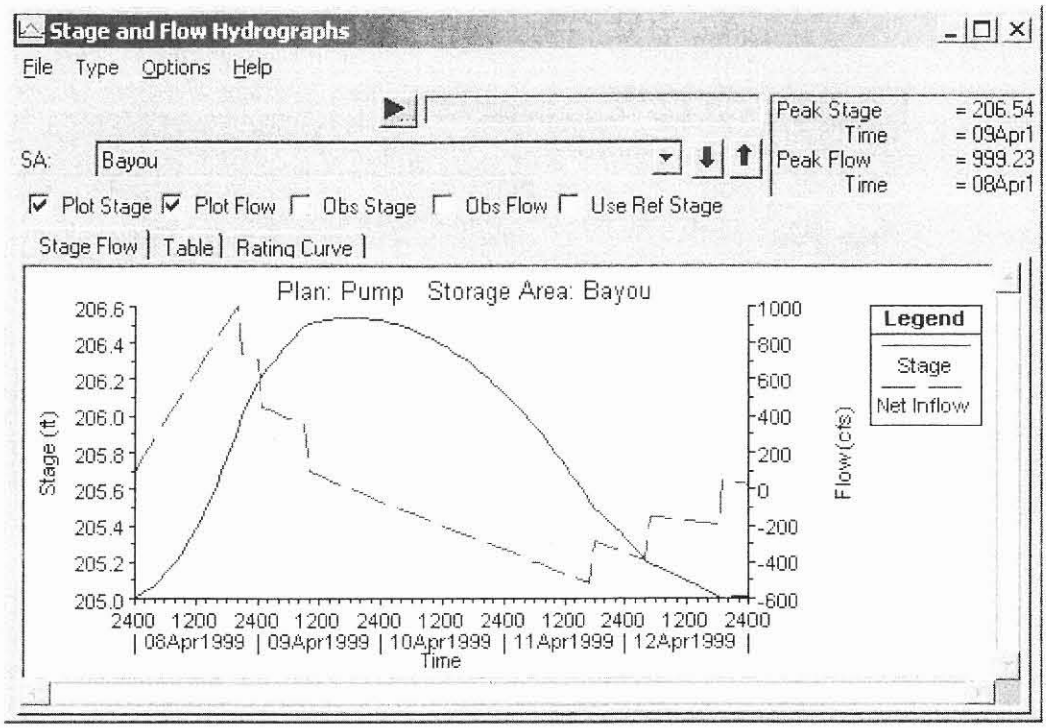


Figure 16.22 Stage and Flow Hydrographs for the Storage Area

Navigation Dams

This section discusses the navigation dam option in HEC-RAS. For a navigation dam, the program will try to maintain both a minimum and maximum water surface in one or more locations along a navigation channel. The program does this by controlling the gate settings on an inline structure. The user enters a target water surface (and various other calibration data) and the program will adjust the gate settings at user specified time intervals in order to meet the target water surface as closely as possible. This section describes the data requirements for a navigation dam and includes a general discussion of how the gate operations are performed.

The first step in modeling a navigation dam is to add the physical data for the navigation dam by selecting the Inline Structure option on the Geometry Data editor and entering the appropriate information. The next step is to add the inline weir as a boundary condition on the Unsteady Flow editor and then click the **Navigation Dams** button. The editor, as shown in Figure 16.23, will appear (note: the fields will be blank when first brought up).

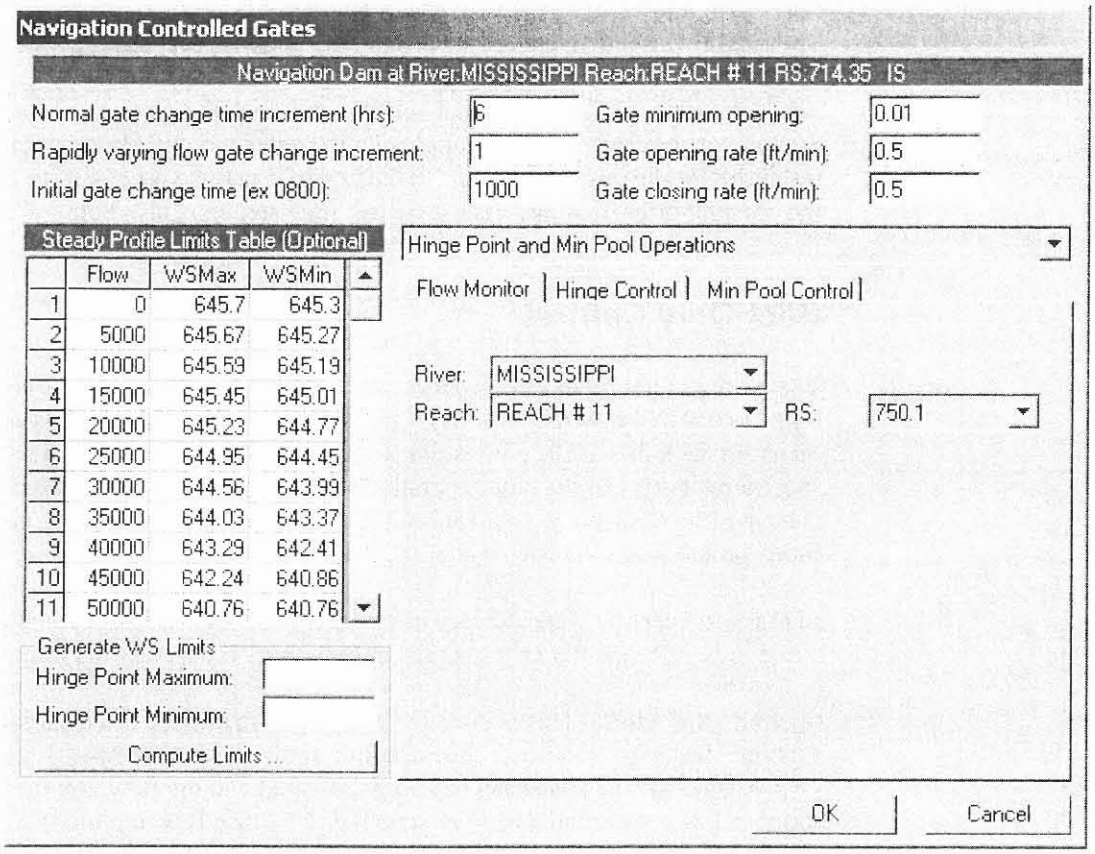


Figure 16.23. Navigation Dam Editor with Flow Monitor

Normal gate change time increment – This field states how often the program will adjust the gate settings. In the example shown in Figure 16.23, the program will only make adjustments to the gates every six hours under normal operations.

Rapidly varying flow change increment – This field represents the minimum length of time between gate setting adjustments. For example, during rapidly changing conditions, the program can adjust the gates up to once an hour in order to maintain the appropriate water surfaces.

Initial gate change time – This field is the military style time for when the first gate change will take place. In this example, it is 10:00am. If the simulation starts after 10:00am then the gates will be first adjusted at 4:00pm, 10:00pm, or 4:00am as appropriate.

Gate minimum opening – This field is the minimum opening for the first gate group (the first gate group as defined on the Geometry editor). The program will keep the gates on this gate group open to at least 0.1 feet. The other gate groups may be closed completely (see discussion of gate opening and closing below).

The final two fields [*Gate opening and Gate closing rate*] are the maximum speed that the gates in any gate group can be opened or closed. Generally this rate is determined by the physical speed with which the gates can be adjusted. Sometimes, however, opening or closing the gates too quickly can cause instability in the unsteady solver. In this case, it may be necessary to reduce the opening or closing rate. But a shorter time step may also help.

Pool Only Control

There are several types of navigation dam operations. The simplest is pool only control (as shown in Figure 16.23). In this case, the program tries to maintain the water surface immediately upstream of the dam within user specified targets. In the other operations (see below), the target water surface is located some distance upstream of the dam and there may or may not be limits on the water surface right at the dam.

In order to keep the water surface at the dam within the user specified limits, while only infrequently changing the gate settings (i.e., every six hours), the program needs to know what the approximate inflow at the dam will be some time into the future. This is done by monitoring the flow at an upstream cross section. The user must enter this location. In the above example (Figure 16.23), the **Flow Monitor** tab has been activated and the flow monitor location has been entered as river station 315.5. The flow monitor location should be chosen so that the river travel time between the monitor location and the navigation dam is on the order of (or somewhat less than) the normal gate increment. In this example (Figure 16.23) the gate time increment is every six hours, so a location a few hours upstream would be appropriate.

The calibration of the navigation dam control data involves some empirical decisions and trial and error experimentation. This is true of the flow monitor location as well as most of the remaining data below.

The flow monitor location must be a normal cross section in the model. This means that cross sections must be extended far enough upstream of the dam to account for this location. At some point, an option may be added for the navigation dam to see into the upstream hydrograph so that there is no minimum requirement of upstream cross section distance. Note also that the monitor point can be located upstream of other hydraulic structures including other navigation dams. As long as another upstream navigation dam does not have a significant storage capacity, it should not affect the results of the flow monitor. The ability of one navigation dam to "talk" to a downstream dam may be added in the future.

After the flow monitor location has been chosen, the **Pool Control** tab can be pressed bringing up the editor shown in Figure 16.24.

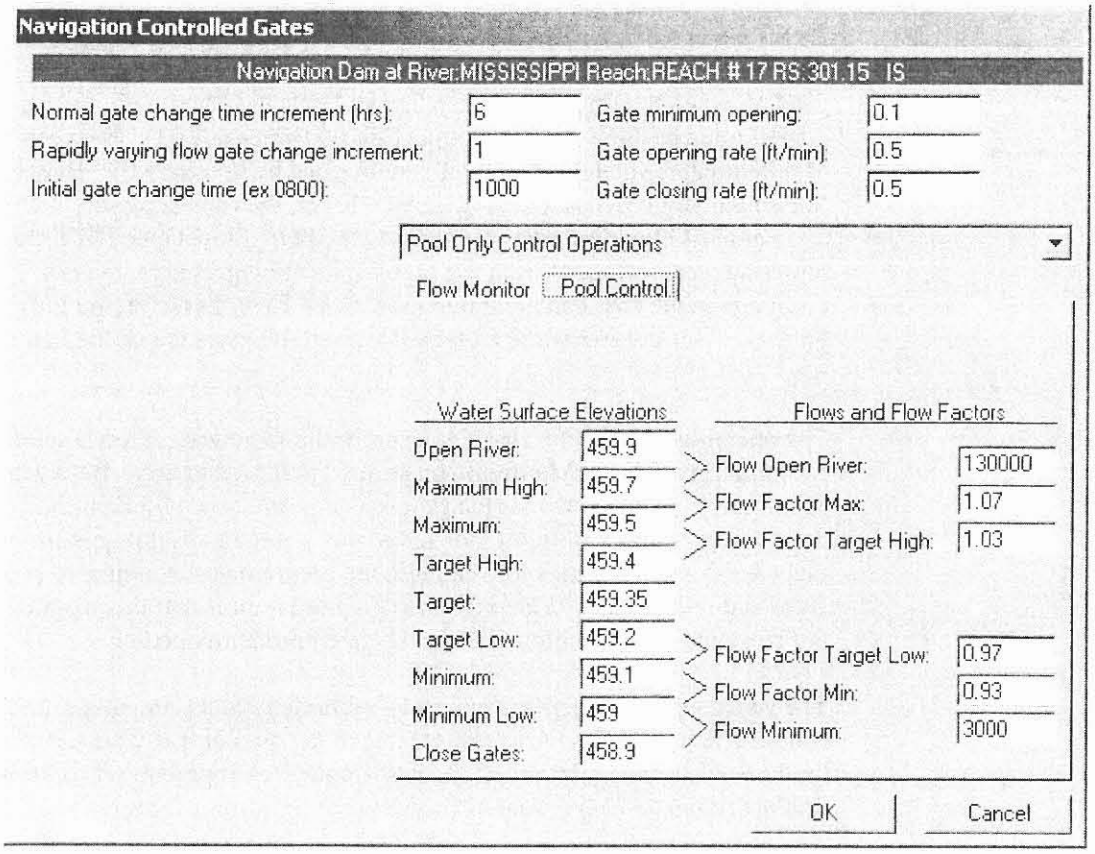


Figure 16.24. Navigation Dam Editor with Pool Control

The user enters a range of water surfaces and corresponding *Flow Factors*. In this example, the ideal target water surface has been entered as 459.35. The primary target range is from 459.2 (Target Low) to 459.5 (Target High). In general, if the water surface is between *Target Low* and *Target High* and it is time to change the gate settings, then the program will adjust the gates to get an average of the current flow at the dam and the monitor flow.

For instance, assume that at time 10:00 the current discharge from the navigation dam is 10,000 cfs, 11,000 cfs of flow is observed at the monitor location, and the water surface at the dam is 459.4 feet. Since 459.4 is in the primary target range, the program will compute the average of the flows, 10,500 cfs. By trial and error, the program will change the gates (and compute the corresponding flow) until there is 10,500 cfs (plus or minus the tolerance) of discharge at the dam. The tolerance is 1% of the flow, in this case 105 cfs. So the program will actually stop iterating whenever it first determines a gate setting that results in a flow that is between 10395 cfs and 10605 cfs. After the gates have been changed, they won't (normally) be adjusted for the next six hours. The flow from the dam will, of course, still vary as the water surface at the dam fluctuates.

As the water surface at the dam gets out of the primary target range, then the flow (that is, the discharge from the dam) is adjusted by the Flow Factors. In general, when the stage is between Target High and Maximum, then the flow is multiplied by Flow Factor Target High (in this case 1.03). Between Maximum and Maximum High, it is multiplied by at least 1.07. Between Maximum High and water surface Open River, the flow is rapidly increased up to at least Flow Open River (listed as 50,000). Flow Open River does not represent a cap. If the flow at the monitor location gets high enough, the discharge at the dam can go above Flow Open River based on the Flow Factors. Above water surface Open River, all the gates are opened all of the way.

The operations below the target zone work the same way. Flow Factor Target Low and Flow Factor Minimum are applied in the same way. Between Minimum Low and water surface Close Gates, the flow will be rapidly decreased to Flow Minimum, but again, this is not an absolute minimum. If the water surface remains low enough, the program will continue to close the gates and reduce flow. The only absolute minimum is that the program will not close the first gate group below the gate minimum opening.

The water surface targets are basically calibration knobs and no particular water surface targets have to exactly match the operationally prescribed limits on the pool surface. However, the best response will probably be obtained if the Maximum and Minimum are close to the prescribed limits.

Hinge Point Only Control

The next type of navigation dam operation is hinge point control. This is similar to pool control. The main difference is that instead of the water surface targets being located right at the face of the dam, the water surface targets are located some distance upstream. Figure 16.25 shows the Hinge Point Only editor. (Hinge point control is selected by clicking on the drop down box near the top right of the editor.)

In this example, the navigation dam is located at river station 714.35, and the hinge point is located at 728.28. The program will adjust the gates at the dam in order to maintain an approximate water surface of 645.55 feet (the target water surface) at river station 728.28. The target water surfaces and Flow Factors behave the same as for pool control. A flow monitor location is still needed. It should be located an appropriate distance upstream of the hinge point. For this dam, it is located a few hours upstream at river station 750.1 (see Figure 16.25).

The Steady Profile Limits Table is an optional feature (see Figure 16.25). It can make the navigation dam operations more robust for rapidly changing flow. It addresses the situation where the water surface for a give flow at the dam diverges significantly from the water surface that would be expected at the dam for a steady state, uniform flow condition.

A typical example is the trailing end of a high flow hydrograph. For instance, the flows at the hinge point and monitor location may have fallen considerably below the Open River condition, but the water surface at the dam is still a little high (compared to the flow). When the program computes a desired flow at the dam, e.g. 10,000 cfs, it adjusts the gates to get this flow. Over the next six hours, however, as the water surface at the dam continues to fall toward a lower equilibrium, the discharge can drop significantly below 10,000. This means that the navigation dam response is either sluggish in returning to the target water surface at the hinge point and/or the gates have to be changed more frequently. This is where the table becomes useful.

The data in this table give the water surfaces at the dam that will produce the target water surfaces at the hinge point for steady state conditions. For this navigation dam, it is desired to keep the water surface at the hinge point between 645.3 and 645.7 feet. If, for instance, there is a long term (steady state) flow of 10,000 cfs between the hinge point and the dam, then maintaining a water surface at the dam of 645.19 feet will result in a water surface of 645.3 feet at the hinge point. Similarly, a water surface of 645.59 at the dam will result in a water surface of 645.7 feet at the hinge point, for the same 10,000 cfs flow. The table can be semi-automatically generated by HEC-RAS. After entering the hinge points limits (inside the Generate WS Limits box) and entering a range of flows (left-most column of the table), the user can click on the **Compute Limits** button and the program will use the steady state run model to fill in the rest of the table.

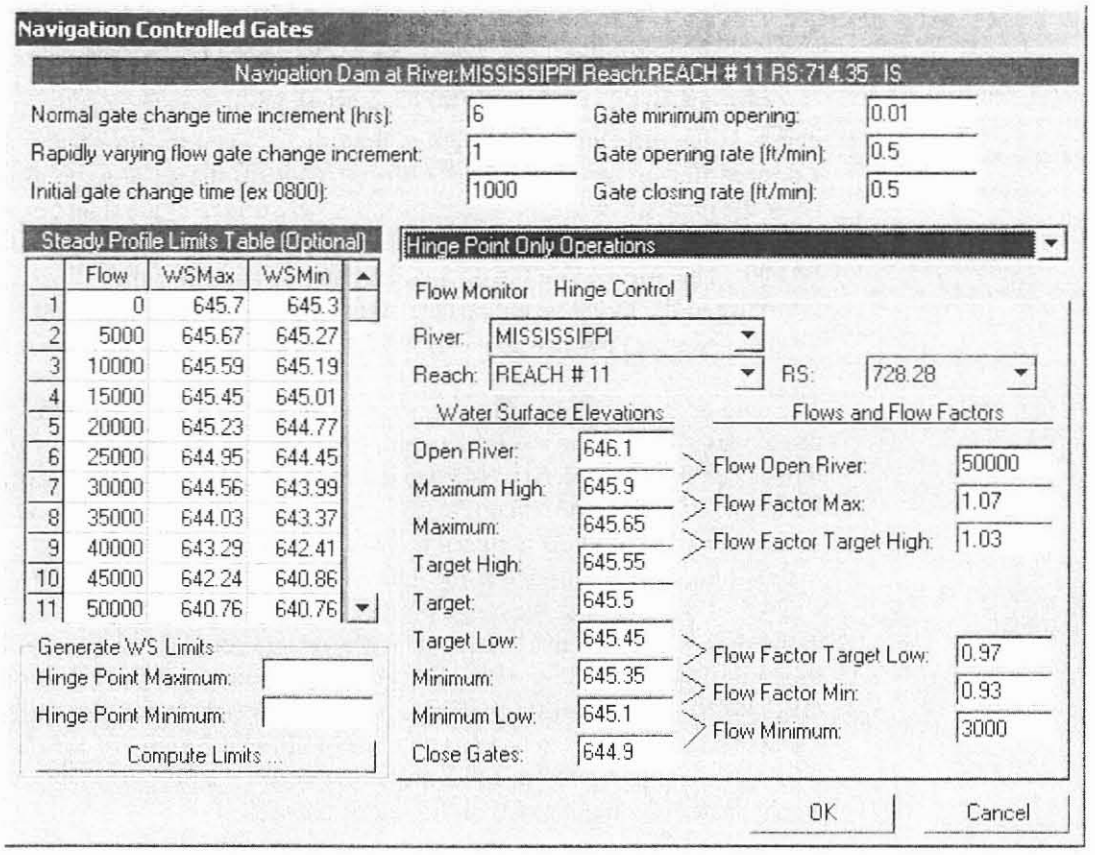


Figure 16.25. Navigation Dam Editor with Hinge Control

Continuing on with the 10,000 cfs flow example, before the program starts to iterate, it checks the current water surface at the dam against the table. If the current water surface is between the limits (in this case 645.19 and 645.59), the program continues normally as it would if the table was not being used (that is, the user had left it blank). However, lets assume that the water surface at the dam is 646.0 feet. This would mean that the water surface at the dam is above the limits. In this case the program will temporarily assume a headwater of 645.59 feet at the dam and determine the gate settings that will result in a discharge of 10,000 cfs for this lower, assumed, headwater. After this has been done, the program will use this gate setting and continue on normally. This will result in the flow at the dam initially being above the 10,000 cfs target. However, as the water surface at the dam drops, the flow should also drop down towards the 10,000 cfs range. This will, hopefully, produce a faster response without over shooting the target water surface at the dam.

If the water surface is on the low side, it works the same way except the lower limit is used. If the water surface at the dam were 645.0 then the gate setting would be based on an assumed water surface of 645.19. The profile table is optional. It can be ignored, by leaving it blank. However, it can produce a better response, at least for some data sets. That being said, it should also be

noted that the table will not perform as well when the flow at the dam is being heavily influenced by tailwater conditions.

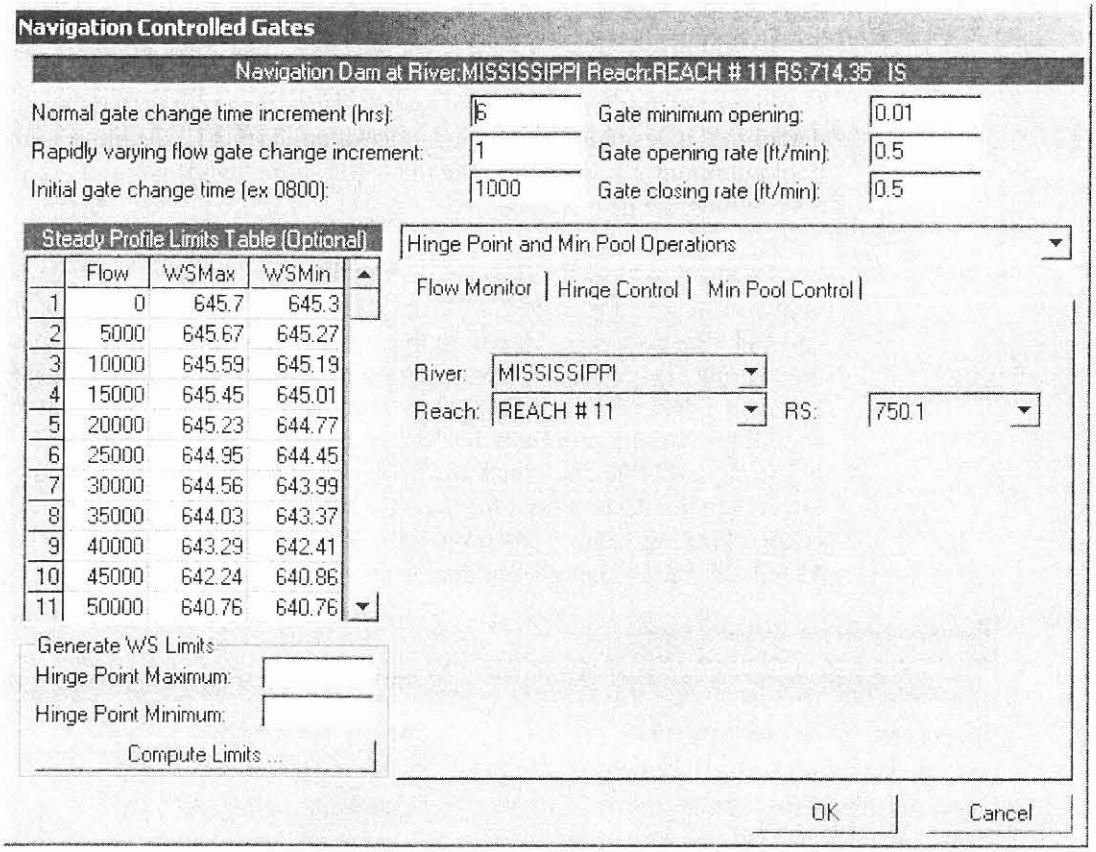


Figure 16.26. Navigation Dam Editor with Steady Profile Limits Table

Hinge Point and Minimum Pool Operations

The hinge point navigation dam operation can also be combined with limits on the water surface at the dam. Hinge point and minimum pool operation will try to maintain the water surface within targets at the hinge point, but only when the water surface at the dam is above certain limits. When the water surface at the dam drops too low, the program will adjust the gates based on the water surface at the dam, essentially reverting to pool only control.

The hinge point and the minimum pool operation are each treated as separate control points. In addition to the water surfaces and Flow Factors for the Hinge control, the pool minimum has its own full set of water surfaces and Flow Factors as shown in Figure 16.27 (these are accessed by clicking on the **Min Pool Control** button). Even though the minimum pool control is only trying to maintain a minimum water surface at the dam, a full range of water surfaces and Flow Factors are needed. These include the "too high" numbers

such as Maximum High and Flow Open River. This allows the program to smoothly transition between hinge control and pool control. It also allows the pool control response to be fully calibrated between sluggish and overly sensitive transitions.

For hinge and pool minimum navigation dams, the program independently determines a desired flow for each control point (that is, the hinge and the pool minimum). It will then take the lower of the two flows and use that for determining the gate settings.

For instance, assume the flows at the monitor location and the hinge point are 40,000cfs and that the water surfaces at the hinge point and the dam are 645.6 and 644.9 respectively. Based on the hinge point conditions (water surface at hinge point, Targets and Flow Factors for the hinge point), the program might compute a desired flow of 41,000cfs. Next the program will look at the conditions, targets, and Flow Factors at the dam and compute a desired flow of, perhaps, 42,000 cfs. Since the desired flow for the hinge point targets is lower than the desired flow for the navigation dam targets, the pool minimum is not a limiting factor. The program will adjust the gate settings to get 41,000 cfs and the navigation dam is operating under hinge control.

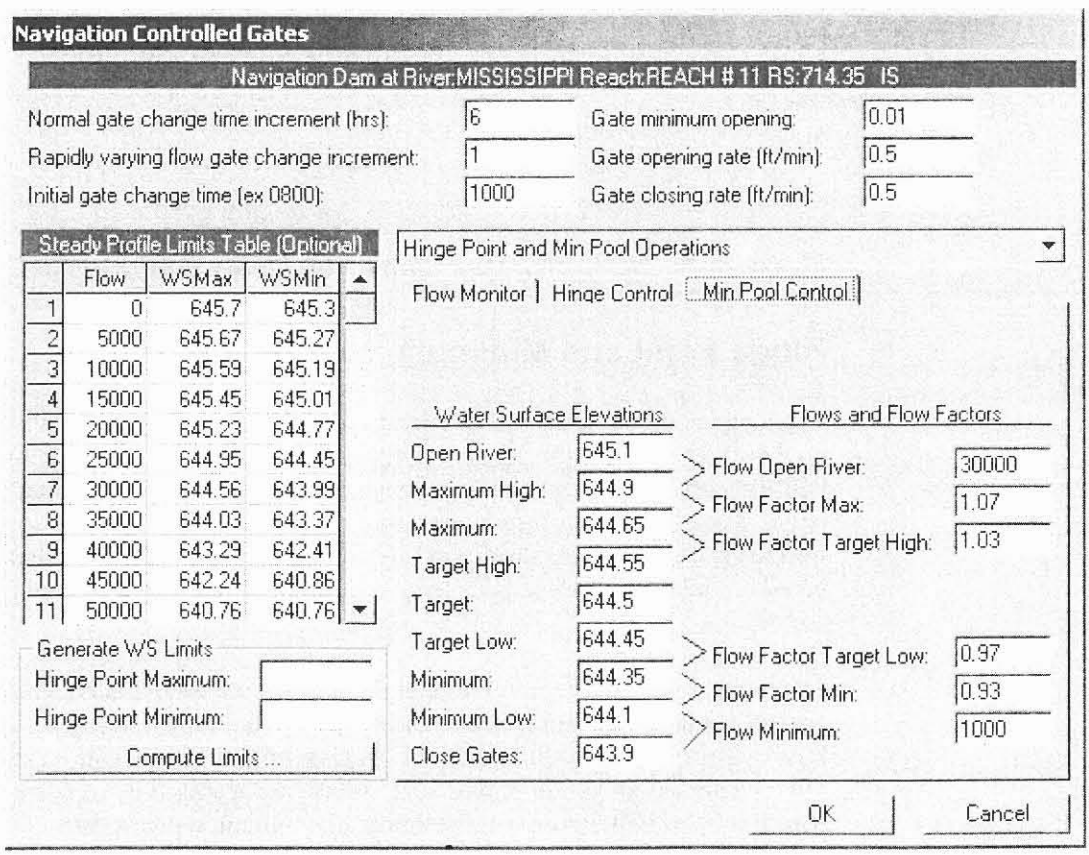


Figure 16.27. Navigation Editor with Hinge Point and Minimum Pool Operations and Control

The next time the gates are adjusted, assume the flow at the monitor and hinge point are still basically 40,000 cfs, but that the water surfaces have dropped to 645.5 feet at the hinge and 644.4 feet at the dam. The new computed flows might be 40,000 cfs at the hinge and 39,000 at the dam. In this case the program would use the 39,000 cfs figure and the dam would be under pool minimum control. In other words, the water surface at the dam has dropped to the point that the program has to operate the gates to maintain a minimum water surface at the dam regardless of what is happening at the hinge point.

The hinge and pool minimum operation is usually under hinge control for low and normal flows. At high flows the water surface at the dam must be lowered in order to keep the hinge point within the target range. At even higher flows, the water surface at the dam cannot be lowered far enough to keep the hinge point in range, thus the dam reverts to pool minimum control. Ideally, the pool would be kept at the specified absolute minimum (perhaps 644.1 feet in the above example) until the hinge point dropped back down into the target range. This is not possible without continuous adjustments of the gates, which is not practicable.

Instead, the water surface at the dam will fluctuate slightly even when it is operating under pool minimum control (just like it would fluctuate for pool only control). This is reflected in the range of target water surfaces for pool minimum control. The spacing of the target water surfaces have to be determined by trial and error. For instance, if the water surface Target, Target High, and so on, are set to relatively high elevations (compared to the desired value), then the water surface at the dam might stay significantly above the minimum of 644.1. This is not desired when the water surface at the hinge point is above the targeted range. Moving the dam target water surfaces closer together (closer to 644.1) will cause the program to increase the flows more quickly in order to drive the water surface back down. However, this can also cause the program to overshoot the desired target leading to frequent gate changes and/or bouncing water surfaces.

If the pool minimum is a hard minimum (a hard minimum might be, the pool should not be allowed to drop below 644.1 feet), then this minimum should be coded as one of the lower target water surfaces. For instance, if 644.1 is the operationally prescribed absolute minimum and the user coded the primary water surface Target as 644.1, then the pool would fluctuate around the value of 644.1 during pool control. It would be better, in this case, to code it to the Minimum Low, for example. On the other hand, if the minimum is a "soft" minimum (a soft minimum might be 644.45 +/- .25 feet) then setting Target Low or even perhaps the primary Target to 644.45 might give better results. As already mentioned, the user should be prepared to take a trial and error approach in order to get the best results.

For hinge point and minimum pool operation, the Steady Profile Limits table can still be optionally used. This table is only used when the dam is operating under hinge control. The water surface values in the table can be lower

elevations than the actual limits on the pool. These values are still used, but the pool control minimum will still apply. For instance, the values in the table go below the 644.1 desired minimum at the pool. During rapidly changing conditions, when the water surface for a given flow diverges from the steady state water surface (for that flow), these lower values can still be used and will (in some cases) give a faster response. However, if the water surface actually drops down to around the 644 to 645 level, the flow based on pool control will eventually be lower than that based on the Hinge/Steady Profile table and the dam will revert to pool control (which, again, does not use the tables).

Hinge Point and Minimum and Maximum Pool Control

The final type of navigation dam operations is combining hinge point control with both a minimum and maximum limit on the water surface at the dam. This editor has a third button as shown in Figure 16.28 and Figure 16.29.

The minimum and maximum pool controls are treated as separate control points even though they are both located immediately upstream of the dam. They each have a full set of target water surfaces and Flow Factors. The program will compute a desired flow for each control point. So there will be a flow based on the hinge point targets, a flow based on the minimum pool elevation, and a flow based on the maximum pool elevation. During normal operations, the flow will be based on the hinge point target. However, the desired flow will not be allowed to go below the minimum pool control flow and it will not be allowed to go above the maximum pool control flow.

Having separate control points for the minimum and maximum control allows a smooth transition between pool control (either high or low) and hinge control for a full range of flows. It also provides the greatest control and sensitivity for allowing the water surface at the pool to be maintained within the tightest tolerances.

The optional steady profile limits table may still be used. As before, it only applies to hinge control.

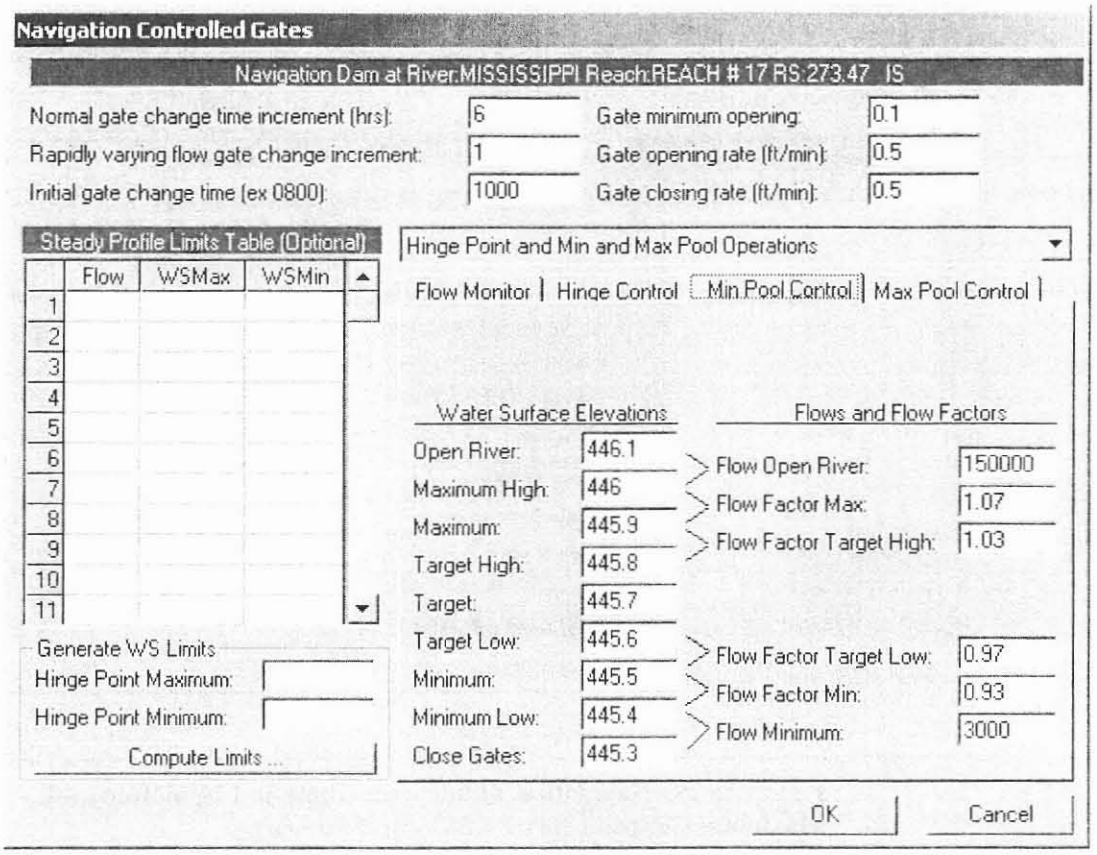


Figure 16.28. Navigation Editor with Hinge and Maximum and Minimum Control (Min Pool Control Shown)

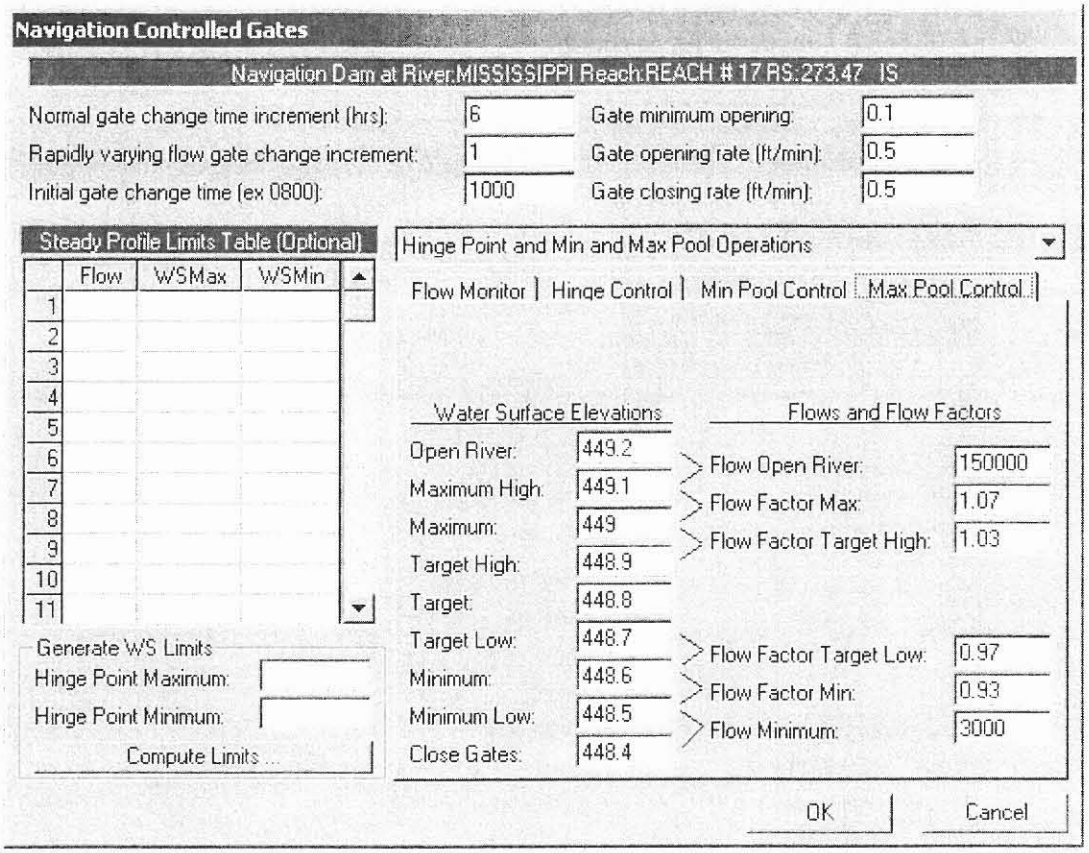


Figure 16.29. Navigation Editor with Hinge and Minimum and Maximum Control (Max Pool Control Shown)

Appendix A References

Barkau, Robert L., 1992. *UNET, One-Dimensional Unsteady Flow Through a Full Network of Open Channels*, Computer Program, St. Louis, MO.

Bureau of Public Roads (BPR), 1965. *Hydraulic Charts for the Selection of Highway Culverts*, Hydraulic Engineering Circular No. 5, U.S. Department of Commerce.

Bureau of Reclamation, 1977. *Design of Small Dams*, Water Resources Technical Publication, Washington D.C..

Bureau of Reclamation, 1998. *Prediction of Embankment Dam Breach Parameters*, Dam Safety Research Report DSO-98-004, Dam Safety Office, U.S. Department of the Interior.

Federal Emergency Management Agency, 1985. *Flood Insurance Study Guidelines and Specifications for Study Contractors*, FEMA 37, Washington D.C., September 1985.

Federal Highway Administration, 1978. *Hydraulics of Bridge Waterways*, Hydraulic Design Series No. 1, by Joseph N. Bradley, U.S. Department of Transportation, Second Edition, revised March 1978, Washington D.C..

Federal Highway Administration, 1985. *Hydraulic Design of Highway Culverts*, Hydraulic Design Series No. 5, U.S. Department of Transportation, September 1985, Washington D.C..

Fread, D.L., 1988 (revised 1991). *BREACH: An Erosion Model For Earth Dam Failures*, National Weather Service, NOAA, Silver Springs Maryland.

Fread, D.L., Ming Jin and Janice M. Lewis, 1996. *An LPI Numerical Solution for Unsteady Mixed Flow Simulation*, North American Water Congress, Anehiem, CA June 22-28, 1996, American Society of Civil Engineers.

FHWA, 1996. *Evaluating Scour at Bridges*, Federal Highway Administration, HEC No. 18, Publication No. FHWA-IP-90-017, 2nd Edition, April 1993, Washington D.C.

Froehlich, D.C., 1988. *Analysis of Onsite Measurements of Scour at Piers*, Proceedings of the ASCE National Hydraulic Engineering Conference, Colorado Springs, CO.

Froehlich, D.C., 1989. *Local Scour at Bridge Abutments*, Proceedings of the 1989 National Conference on Hydraulic Engineering, ASCE, New Orleans, LA, pp. 13-18.

Froehlich, D.C., 1995. *Embankment Dam Breach Parameters Revisited*, Water Resources Engineering, Proceedings of the 1995 ASCE Conference on Water Resources Engineering, San Antonio, Texas, August 14-18, 1995.

Hydrologic Engineering Center, 1991. *HEC-2, Water Surface Profiles*, User's Manual, U.S. Army Corps of Engineers, Davis CA.

Hydrologic Engineering Center, 1993. *UNET, One-Dimensional Unsteady Flow Through a Full Network of Open Channels*, User's Manual, U.S. Army Corps of Engineers, Davis, CA.

Hydrologic Engineering Center, 1994. HECDSS, User's Guide and Utility Programs Manual, U.S. Army Corps of Engineers, Davis CA.

Hydrologic Engineering Center, 1995. RD-41, A Comparison of the One-Dimensional Bridge Hydraulic Routines from: HEC-RAS, HEC-2, and WSPRO, U.S. Army Corps of Engineers, Davis CA., September 1995

Hydrologic Engineering Center, 1995. RD-42, Flow Transitions in Bridge Backwater Analysis, U.S. Army Corps of Engineers, Davis CA., September 1995

Laursen, E.M., 1960. *Scour at Bridge Crossings*, ASCE Journal of Hydraulic Engineering, Vol. 89, No. HY 3.

Laursen, E.M., 1963. *An Analysis of Relief Bridges*, ASCE Journal of Hydraulic Engineering, Vol. 92, No. HY 3.

McDonald, Thomas C. and Jennifer Langridge-Monopolis, 1984. *Breaching Characteristics of Dam Failures*, Journal of Hydraulic Engineering, vol. 110, No. 5, pp 567-586.

Microsoft Corporation, 1998. *Microsoft Windows 95*, User's Manual, Redmond WA.

Richardson, E.V., D.B. Simons and P. Julien, 1990. *Highways in the River Environment*, FHWA-HI-90-016, Federal Highway Administration, U.S. Department of Transportation, Washington, D.C.

U.S. Army Corps of Engineers, 1965. *Hydraulic Design of Spillways*, EM 1110-2-1603, Plate 33.

Files for Geospatial Data

Appendix B

HEC-RAS Import/Export Files for Geospatial Data

At version 2.0, HEC-RAS has introduced three-dimensional (3D) geometry for the description of river networks and cross-sections. This capability makes it possible to import channel geometry from CADD or GIS programs without conversion from real-world coordinates to station-elevation descriptions for the cross sections, as HEC-2 required. Similarly, water-surface elevations calculated at cross sections can be exported to CADD or GIS programs, where they can be used to create model water surfaces for inundation mapping.

Supported HEC-RAS Data Exchange

Using a formatted ASCII text file, HEC-RAS will import a basic description of the channel geometry including:

- The structure of the stream network, as represented by interconnected reaches.
- The location and description of cross sections.
- Manning's *n* values for each cross section.
- Levee alignment information for each cross section.
- Ineffective flow area locations for each cross section.
- Storage area information for each cross section.

Using the same file format, HEC-RAS can write a file exporting the results of a hydraulic model run to a CADD or GIS program. At a minimum, reported results include the locations of cross sections and the calculated water-surface elevations at those cross sections.

The Import/Export Data File Structure

This section gives general rules for the construction of an HEC-RAS geometric data import or export file. It is not necessary to understand all these rules to build an import file, but they may be useful when debugging failed imports. The rules given here are a portion of the definition of a general-purpose geometric data exchange format being developed at HEC for its NexGen model programs. **Note: These file formats are evolving,**

in that additional data types will be added, and some of the existing ones may be modified for future versions. If you are writing software to read and write these file formats, please keep in mind that you may need to modify your software to stay compatible with future versions of HEC-RAS.

Records and Keywords

The HEC-RAS geometric data import file is composed of records, which in turn are composed of keywords and values. All records must contain one keyword, and all keywords end with a colon (:). A record can also contain a value or a set of values following the keyword, i.e., after the colon. Spaces, tabs, or line ends can be placed between a keyword and values within a record.

A record that contains a keyword and no value marks the beginning or the end of a group of related records (for example, the record "BEGIN HEADER:" marks the beginning of the header section of a data file). A record that contains a keyword and a value assigns that value to the part of the model named by the keyword.

When a keyword is read, all spaces up to the colon are removed and all letters are capitalized. The keywords "Begin Header:", "Begin header:", and " Be GiNH eadEr:" are all equivalent to "BEGINHEADER:". For readability, keywords named in this manual will contain internal spaces.

Values

A record can assign a single value to a single variable, or multiple values to an array. Values can be integers, floating point numbers, text strings, or locations (X,Y,Z, label). A single value in an array of values is called an "element" of that array.

A **numerical value (integer or floating point)** cannot contain internal blanks. A floating point number can contain a decimal point; an integer cannot. Elements in an array of numerical values can be separated by commas, blanks, tabs, or line ends.

A **text string** can contain internal blanks, tabs, and commas, but cannot contain internal line ends. Elements in an array of text strings must be separated by line ends.

A **location** consists of three coordinate values and a label (X, Y, Z, label). The first two coordinates are planar, the third gives elevation. The coordinate values are floating point numbers, and the label can be any type of value (although the label can be restricted to a particular data type

in a particular context). In certain contexts, the elevation value or the label may not be required. If a label is used, all three coordinate values must be given; the value "NULL" is valid for the elevation coordinate only. The coordinate values and the label can be separated by commas, blanks, or tabs, but a location cannot contain internal line ends. Elements in an array of locations must be separated by line ends.

Data Groups

Records in the data file can be collected in two types of groups: objects and file sections. An object is a group of records that combine to describe an entity within the model, a cross-section for example. A file section is a logical or functional grouping of data, the file header, for example, is a section that contains a description of the whole file.

Objects and file sections begin and end with records that contain keywords, but no values. A file section starts with a record containing a keyword composed of the word "BEGIN" followed by the section name and a colon, and ends with a keyword composed of the word "END" followed by the section name and a colon. For example, records containing only the keywords "BEGIN HEADER:" and "END HEADER:" are used to start and end the header section of a file. An object starts with a record containing a keyword naming the object type and ends with a record containing the keyword "END:" only. For example, a cross-section object begins and ends with records containing the keywords "CROSS-SECTION:" and "END:" only.

Comments

Hash characters (#) are used to identify comments. When a hash character is encountered in the file, all data from the hash to the next line end are ignored. A line that begins with a hash is equivalent to a blank line.

HEC-RAS Channel Geometry Import File

HEC-RAS reads channel geometry from a text file composed of three data sections:

1. A header containing descriptions that apply to all data in the file.
2. A description of the stream network identifying reach locations and connectivity.
3. A description of the model cross-sections containing their location on the stream network and data required to support the HEC-RAS model.

An example HEC-RAS Channel Geometry Import file and HEC-RAS

model results export file is shown at the end of this appendix.

Header

The header is bounded by the records "BEGIN HEADER:" and "END HEADER:" and must contain a record to identify the units system used in the imported data set. The units system can be ENGLISH or METRIC.

```
BEGIN HEADER:
```

```
UNITS: ENGLISH
END HEADER:
```

Records that may be included in the header are listed in the Table B.1:

Table B.1

Keyword	Value Type	Value
UNITS:	string	ENGLISH or METRIC
PROFILES:	string array	List of profiles exported from HEC-RAS. Not used on import.
DTM TYPE:	string	type (e.g., TIN or raster)
DTM:	string	name of digital terrain model
STREAM LAYER:	string	name of stream layer in CADD or GIS
NUMBER OF REACHES	integer	number of hydraulic reaches contained in the file.
CROSS-SECTION LAYER:	string	name of cross-section layer in CADD or GIS
NUMBER OF CROSS-SECTIONS:	integer	number of cross sections in the file
MAP PROJECTION:	string	projection (coordinate) system used (e.g., STATEPLANE)
PROJECTION ZONE:	string	projection zone (if applicable, e.g., 5101)
DATUM:	string	reference datum for planar coordinates
VERTICAL DATUM:	string	reference datum for vertical coordinates

Stream Network

The stream network section is bounded by the records "BEGIN STREAM NETWORK:" and "END STREAM NETWORK:" and contains records describing reaches and reach endpoints. At a minimum, the stream network section must contain at least two endpoints and one reach. The minimum requirements for a stream network are shown below.

```
BEGIN STREAM NETWORK:
   ENDPOINT:  476132.66, 65291.86, 155.28, 1
   ENDPOINT:  478144.53, 64296.61, 123.72, 2

   REACH:
      STREAM ID: Below Springfield
      REACH ID: Blue River
      FROM POINT: 1
      TO POINT: 2
      CENTERLINE:
         476132.66, 65291.86, 155.28, 23.13
         476196.08, 65196.61, 154.47, 23.09
            lines omitted
         478144.53, 64296.61, 123.72, 22.41
   END:
END STREAM NETWORK:
```

A reach endpoint is represented by a record containing the keyword "ENDPOINT:" followed by four comma-delimited fields containing the endpoints X,Y,Z coordinates and an integer ID.

A reach is represented by a multi-record object that begins with a record containing only the keyword "REACH:" and ends with a record containing only the keyword "END:". At a minimum, a reach object must contain records setting values for a stream ID, a reach ID, a FROM point, and a TO point. A reach's FROM and TO point IDs must match IDs for endpoints listed before the reach object in the file. The reach object must also contain an array of locations defining the stream centerline. This array begins with a record containing only the keyword "CENTERLINE:" and ends when any keyword is encountered. A location element in the array contains the X, Y, and Z coordinates of a point on the stream centerline, and the point's river station. In HEC-RAS, elevation and stationing are optional in the stream network definition. If a location element includes a station value, it must occupy the fourth field in the element. If the elevation is not known, the word "null" must take its place.

Station values are assumed to be in miles for data sets in English units, and in kilometers for data sets in metric units. Stationing is used for indexing locations along reaches, and is not used to precisely locate objects in the model.

Records that may be included in a stream network section are listed in Table B.2:

Table B.2

Keyword	**Value Type**	**Value**
ENDPOINT:	location	coordinates and integer ID
REACH:	none	marks beginning of reach object
END:	none	marks end of reach object
The following records are required for a reach object.		
STREAM ID:	string	identifies reach's membership in stream
REACH ID:	string	unique ID for reach within stream
FROM POINT:	string	integer reference to upstream endpoint
TO POINT:	string	integer reference to downstream endpoint
CENTERLINE:	location array	array elements contain coordinates and (optionally) floating point station value.

Cross Sections

The cross-sectional file section begins with a record containing the only the keyword "BEGIN CROSS-SECTIONS:" and ends with a record containing the only the keyword "END CROSS-SECTIONS:." A cross section is represented by multi-record object beginning with a record containing only the keyword "CROSS-SECTION:" and ending with a record containing only the keyword "END:."

A cross-sectional object must include records identifying the stream, reach, and station value of the cross-section, a 2D cut line, and a series of 3D locations on the cross section. A cut line is composed of the label "CUT LINE:" followed by an array of 2D locations. A cross-sectional polyline consists of the label "SURFACE LINE:" plus 3D coordinates written as comma-delimited X,Y, Z real-number triples, one triple to a line.

Records that may be included in the cross-section file section are listed in Table B.3:

Table B.3

Keyword	**Value Type**	**Value**
CROSS-SECTION:	none	marks beginning of cross-section object
END:	none	marks end of cross-section object
The following records are required for a cross-section object.		
STREAM ID:	string	identifiers for stream and reach where cross-section is located (must refer to existing streams and reaches in the model)
REACH ID:	string	
STATION:	floating point	relative position of cross-section on stream
CUT LINE:	location array	array elements contain 2D coordinates of cross sectione stike line
SURFACE LINE:	location array	array elements contain 3D coordinates of cross section points
The following records are optional for a cross-section object.		
BANK POSITIONS:	floating point (2 elements)	Fraction of length along cut line where main channel bank stations are located. (values 0.0 - 1.0)
REACH LENGTHS:	floating point (3 elements)	Distance along left overbank, center channel, and right overbank flow paths to next cross-section downstream (units are feet or meters).
NVALUES:	floating point (n paired elements)	Manning's n values expressed as fraction along cut line to start of n-value *(fraction, n-value)*.

Keyword	Value Type	Value
LEVEE POSITIONS:	mixed array elements	Levee positions expressed as a unique ID, fraction along the cut line and levee elevation *(Levee_ID, fraction, elevation)*
LEVEE ID:	integer	Unique integer value identifying a levee
INEFFECTIVE POSITIONS:	floating point (paired element arrays)	Ineffective flow area information along the cut line is specified by a unique ID, beginning and ending fraction along the cut line, and elevation *(Ineffective_ID, BeginPct, EndPct, Elevation)*
INEFFECTIVE ID:	integer	Unique integer value identifying ineffective flow area.
SA ID:	integer	Unique integer value identifying a storage area.
POLYGON:	floating point (n paired elements)	Location of ineffective flow areas or storage areas used for plotting.
ELEVATION-VOLUME:	floating point (n paired elements)	Elevation-volume information for a storage area *(elevation,volume)*.
TERRAIN:	floating point (n x,y,z elements)	Elevation data describing the land surface at a storage area *(X,Y,Z)*.
WATER ELEVATION:	floating point array	Water surface elevation values. Used for export of model results. Not read on import.

HEC-RAS Model Results Export File

HEC-RAS exports model results to a text file using the same format as the data import file. The contents of the files, however, are not identical. The stream network section is not required for data export, and the surface line may be omitted from the cross-section objects. An example HEC-RAS model export file is shown at the end of this discussion. Model results are reported with the following elements (Table B.4), which are not required (and are not read) in the import file.

Table B.4

Keyword	**Value Type**	**Value**
The following record is optional in the Header section of the export file.		
PROFILE NAMES:	string array	name(s) of water surface profiles reported in the file. This record is required if more than one profile is reported.
The following record is required for each cross-section object.		
WATER ELEVATIONS:	floating point array	Elevation of water surface at the cross-section. The array must contain one value for each profile.
The following records are optional for a cross-section object.		
PROFILE ID:	string	Water surface profile name. This must match a name in the Profile Names record in the header.
VELOCITIES:	floating point (pair)	Fraction along cut line and value of velocity (*fraction, value*). *Velocities record must follow Profile ID record.*
The following records make up a section defining a bounding polygon of the water surface limits.		
BEGIN BOUNDARIES:	none	Marks start of boundaries file section.
END BOUNDARIES:	none	Marks end of boundaries file section.
PROFILE LIMITS:	none	Marks start of an object defining the limits of a single water surface profile.
PROFILE ID:	string	Name of profile. This must match a name in the Profile Names record in the header.
POLYGON	location array	A series of 2D locations marking the limits of a water surface. A single profile limit can be merged from multiple polygons.

1. HEC-RAS allows the user 16 character profile names. However, **profile names can contain up to 9 characters for HEC-GeoRAS ArcView Extension Version 3.0 or 11 characters for HEC-GeoRAS for ArcInfo.** They must begin with a letter.

2. If no profile name is provided, only one water elevation will be written for each cross section.

Water Surface Bounding Polygon

In addition to a water surface elevation at each cross section (one for each profile), the HEC-RAS program sends a bounding polygon for each hydraulic reach in the model (the program outputs a new set of bounding polygons for each profile computed). The bounding polygon is used as an additional tool in assisting the GIS (or CADD) software to figure out the boundary of the water surface on top of the terrain.

In most cases, the bounding polygon will represent the outer limits of the cross section data, and the actual intersection of the water surface with the terrain will be inside of the polygon. In this case, the GIS software will use the water surface elevations at each cross section and create a surface that extends out to the edges of the bounding polygon. That surface is then intersected with the terrain data, and the actual water limits are found as the location where the water depth is zero.

However, is some cases, the bounding polygon may not represent the extents of the cross-section data. For example, if there are levees represented in the HEC-RAS model, which limit the flow of water, then the bounding polygon will only extend out to the levees at each cross section. By doing this, when the information is sent to the GIS, the bounding polygon will prevent the GIS system from allowing water to show up on both sides of the levees.

In addition to levees, the bounding polygon is also used at hydraulic structures such as bridges, culverts, weirs, and spillways. For example, if all of the flow is going under a bridge, the bounding polygon is brought into the edges of the bridge opening along the road embankment on the upstream side, and then back out to the extent of the cross-section data on the downstream side. By doing this, the GIS will be able to show the contraction and expansion of the flow through the hydrualic structures, even if the hydraulic structures are not geometricaly represented in the GIS.

Another application of the bounding polygon is in FEMA floodway studies. When a floodway study is done, the first profile represents the existing conditions of the flood plain. The second and subsequent profiles are run by encroaching on the floodplain until some target increase in water surface elevation is met. When the encroached profile is sent to the GIS, the bounding polygon is set to the limits of the encroachment for each cross section. This will allow the GIS to display the encroached water surface (floodway) over the terrain, even though the water surface does not intersect the ground.

Import/Export Guidelines

The following rules apply to channel and cross-section import/export data.

Defining The Stream Network

1. The stream network is represented by a set of interconnected reaches. A stream is a set of one or more connected reaches that share a common stream ID.

2. A stream is composed of one or more reaches with the same stream ID, and each reach in a stream must have a unique reach ID. Every reach must be identified by a unique combination of stream and reach IDs.

3. Stream IDs and Reach IDs are alphanumeric strings up to 16 characters long. Reach endpoint IDs are integers.

4. Streams cannot contain parallel flow paths. (If three reaches connect at a node, only two can have the same stream ID.) This prevents ambiguity in stationing along a stream.

5. A reach is represented by an ordered series of 3D coordinates, and identified by a stream ID, a reach ID, and IDs for its endpoints.

6. A reach endpoint is represented by its 3D coordinates and identified by an integer ID.

7. Reaches are not allowed to cross, but can be connected at their endpoints (junctions) to form a network.

8. The normal direction of flow on a reach is indicated by the order of its endpoints. One point marks the upstream or "from" end of the reach, the other marks the downstream or "to" end of the reach.

Defining Cross Sections

1. Each cross section is defined by a series of 3D coordinates, and identified by a stream name and reach name (which must refer to an existing stream and reach) and a station, indicating the distance from the cross-section to the downstream end of the stream.

2. Careful attention must be given to cross-sectional stationing.

3. A cross-section line can cross a reach line exactly once, and cannot

cross another cross-section line.

Results of a water surface calculation are exported in a file that contains cross-section locations in plane (2D) coordinates, water-surface elevations for the cross-sections, and boundary polygons for the reaches.

The Following Rules Apply to Water-Surface Export Data

1. A cross-section is represented by a water surface elevation and a series of 2D coordinates on the cross-section cut line. The full width of the cross-section is included.

2. One bounding polygon is created for each reach in the stream network, and for each profile.

3. A reach's bounding polygon is made up of the most upstream cross-section on the reach, the endpoints of all cross-sections on the reach, and the most upstream cross-sections of reaches downstream of the reach.

4. For purposes of defining bounding polygons *only*, the endpoints of a cross-section are adjusted to the edge of the water surface at the cross-section if the cross-section is part of a floodway, a leveed section of the reach, or the water extent is controlled by a hydraulic structure. This allows calculated water surfaces that are higher than the land surface to be reported back to the CADD or GIS program.

Sample HEC-RAS Geometry Import File

```
# RAS input file created on Thu Nov 18 13:12:37 1999
# by ArcView extension HEC-GeoRAS

BEGIN HEADER:

  DTM TYPE: TIN
  DTM: d:\georas\wailupe\wai_tin
  STREAM LAYER: d:\georas\wailupe\stream3d.shp
  NUMBER OF REACHES: 3
  CROSS-SECTION LAYER: d:\georas\wailupe\xscutlines3d.shp
  NUMBER OF CROSS-SECTIONS: 40
  MAP PROJECTION: STATEPLANE
  PROJECTION ZONE: 5103
  DATUM: NAD27
  UNITS: ENGLISH

END HEADER:


BEGIN STREAM NETWORK:

  ENDPOINT: 582090.487, 49258.898, 218.609, 1
  ENDPOINT: 582331.707, 47063.536, 114.164, 2
  ENDPOINT: 583735.405, 47715.344, 278.222, 3
  ENDPOINT: 584138.295, 41249.225, 1.140, 4

  REACH:
    STREAM ID: Wailupe
    REACH ID: Upper
    FROM POINT: 1
    TO POINT: 2
    CENTERLINE:
      582090.487, 49258.898, 218.609, 8640.151
     many lines omitted
      582331.707, 47063.536, 114.164, 6402.057
  END:

  REACH:
    STREAM ID: Kulai Gorge
    REACH ID: Tributary
    FROM POINT: 3
    TO POINT: 2
    CENTERLINE:
      583735.405, 47715.344, 278.222, 1813.116
     many lines omitted
      582331.707, 47063.536, 114.164, -0.000
  END:

  REACH:
    STREAM ID: Wailupe
    REACH ID: Lower
    FROM POINT: 2
    TO POINT: 4
```

```
    CENTERLINE:
      582331.707, 47063.536, 114.164, 6402.057
     many lines omitted
      584138.295, 41249.225, 1.140, 0.000
  END:

END STREAM NETWORK:


BEGIN CROSS-SECTIONS:

  CROSS-SECTION:
    STREAM ID: Wailupe
    REACH ID: Lower
    STATION: 220.827
    BANK POSITIONS: 0.503, 0.515
    REACH LENGTHS: 87.418, 220.827, 159.365
    NVALUES:
      0.000, 0.150
      0.501, 0.035
      0.532, 0.150
    LEVEE POSITIONS:
      1, 0.312, 6.234
    INEFFECTIVE POSITIONS:
      1, 0.000, 0.478, 5.789
    CUT LINE:
      586214.122, 42127.918
      581980.991, 40806.059
    SURFACE LINE:
      586214.122, 42127.918, 4.007
     many lines omitted
      581980.991, 40806.059, 6.390
  END:

  CROSS-SECTION:
    STREAM ID: Wailupe
    REACH ID: Lower
    STATION: 346.249
    BANK POSITIONS: 0.506, 0.518
    REACH LENGTHS: 128.209, 125.422, 199.366
    NVALUES:
      0.000, 0.066
      0.021, 0.150
      0.503, 0.035
      0.532, 0.150
    LEVEE POSITIONS:
      1, 0.345, 7.123
    INEFFECTIVE POSITIONS:
      1, 0.000, 0.412,5.987
    CUT LINE:
      586190.939, 42248.509
      583923.687, 41543.517
      581883.624, 41116.812

    SURFACE LINE:
      586190.939, 42248.509, 13.293
```

```
   many lines omitted
     581883.624, 41116.812, 7.145
 END:

many cross sections omitted

 CROSS-SECTION:
   STREAM ID: Wailupe
   REACH ID: Lower
   STATION: 6269.258
   BANK POSITIONS: 0.524, 0.569
   REACH LENGTHS: 170.100, 164.521, 158.965
   NVALUES:
     0.00, 0.066
     0.19, 0.150
     0.47, 0.035
     0.56, 0.150
     0.94, 0.066
   CUT LINE:
     582723.174, 46846.449
     582426.438, 46878.916
     581953.514, 47082.992
   SURFACE LINE:
     582723.174, 46846.449, 161.917
    many lines omitted
     581953.514, 47082.992, 165.010
 END:

 CROSS-SECTION:
   STREAM ID: Wailupe
   REACH ID: Upper
   STATION: 6822.378
   BANK POSITIONS: 0.492, 0.555
   REACH LENGTHS: 142.689, 139.905, 126.201
   NVALUES:
     0.00, 0.066
     0.27, 0.150
   CUT LINE:
     582774.257, 47502.740
     582593.433, 47493.464
     582343.062, 47465.635
     582083.418, 47502.740
   SURFACE LINE:
     582774.257, 47502.740, 170.548
    many lines omitted
     582083.418, 47502.740, 164.059
 END:

  many cross sections omitted

 CROSS-SECTION:
   STREAM ID: Wailupe
   REACH ID: Upper
   STATION: 6682.474
   BANK POSITIONS: 0.380, 0.472
   REACH LENGTHS: 454.362, 413.216, 375.704
```

```
    NVALUES:
      0.00, 0.150
    CUT LINE:
      582641.922, 47366.533
      582444.447, 47335.449
      582336.567, 47337.277
      582062.296, 47381.161
    SURFACE LINE:
      582641.922, 47366.533, 149.810
     many lines omitted
      582062.296, 47381.161, 160.799
  END:

CROSS-SECTION:
    STREAM ID: Kulai Gorge
    REACH ID: Tributary
    STATION: 1089.584
    BANK POSITIONS: 0.373, 0.579
    REACH LENGTHS: 263.179, 255.877, 223.864
    NVALUES:
      0.00, 0.150
      0.48, 0.055
      0.59, 0.150
      0.70, 0.066
    CUT LINE:
      583337.968, 47187.952
      583207.930, 47327.062
      583153.496, 47381.496
      583126.279, 47608.306
    SURFACE LINE:
      583337.968, 47187.952, 257.736
     many lines omitted
      583126.279, 47608.306, 326.921
  END:

  CROSS-SECTION:
    STREAM ID: Kulai Gorge
    REACH ID: Tributary
    STATION: 273.138
    BANK POSITIONS: 0.541, 0.655
    REACH LENGTHS: 139.815, 273.138, 79.293
    NVALUES:
      0.00, 0.150
      0.37, 0.055
      0.62, 0.035
      0.64, 0.150
    CUT LINE:
      582546.842, 47088.605
      582555.984, 47189.171
      582550.499, 47240.368
      582552.327, 47295.223
    SURFACE LINE:
      582546.842, 47088.605, 145.787
    many lines omitted
      582552.327, 47295.223, 144.778
  END:
```

```
END CROSS-SECTIONS:

BEGIN LEVEES:

  LEVEE ID: 1
  SURFACE LINE:
      584579.800, 41808.166, 7.222
   many lines omitted
      584631.334, 41572.921, 5.922
  END:

END LEVEES:

BEGIN INEFFECTIVE AREAS:

  INEFFECTIVE ID: 1
  POLYGON:
      584422.114, 41883.166
      584452.132, 41787.626
   many lines omitted
      584552.852, 41577.321
  END:

END INEFFECTIVE AREAS:

BEGIN STORAGE AREAS:

  SA ID: 1
  POLYGON:
      581919.014, 43565.358
   many lines omitted
      581895.224, 43443.864
  END:

  ELEVATION-VOLUME:
      20.000, 0
      22.000, 675000
   many lines omitted
      30.000, 421300
  END:

  TERRAIN:
      581898.124, 43566.478, 33.442
      582361.222, 43216.332, 20.369
   many lines omitted
      581867.484, 43432.612, 33.356
  END:

END STORAGE AREAS:
```

Sample HEC-RAS Geographic Data Export File

```
BEGIN HEADER:
  UNITS: ENGLISH
  DTM TYPE: TIN
  DTM: d:\georas\wailupe\wai_tin
  STREAM LAYER: d:\georas\wailupe\stream3d.shp
  CROSS-SECTION LAYER: d:\georas\wailupe\xscutlines3d.shp
  MAP PROJECTION: STATEPLANE
  PROJECTION ZONE: 5103
  DATUM: NAD27
  VERTICAL DATUM:
  NUMBER OF PROFILES:  3
  PROFILE NAMES:
    Big
    Bigger
    Biggest
  NUMBER OF REACHES:  3
  NUMBER OF CROSS-SECTIONS:  103
END HEADER:

BEGIN STREAM NETWORK:

  ENDPOINT: 582090.487, 49258.898, 218.609, 1
  ENDPOINT: 582331.707, 47063.536, 114.164, 2
  ENDPOINT: 583735.405, 47715.344, 278.222, 3
  ENDPOINT: 584138.295, 41249.225, 1.140, 4

  REACH:
    STREAM ID: Wailupe
    REACH ID: Upper
    FROM POINT: 1
    TO POINT: 2
    CENTERLINE:
      582090.487, 49258.898, 218.609, 8640.151
     many lines omitted
      582331.707, 47063.536, 114.164, 6402.057
  END:

  REACH:
    STREAM ID: Kulai Gorge
    REACH ID: Tributary
    FROM POINT: 3
    TO POINT: 2
    CENTERLINE:
      583735.405, 47715.344, 278.222, 1813.116
     many lines omitted
      582331.707, 47063.536, 114.164, -0.000
  END:

  REACH:
    STREAM ID: Wailupe
    REACH ID: Lower
    FROM POINT: 2
```

```
    TO POINT: 4
    CENTERLINE:
      582331.707, 47063.536, 114.164, 6402.057
     many lines omitted
      584138.295, 41249.225, 1.140, 0.000
  END:

END STREAM NETWORK:

BEGIN CROSS-SECTIONS:

  CROSS-SECTION:
    STREAM ID: Wailupe
    REACH ID: Upper
    STATION: 8032.371
    CUT LINE:
      582496.067, 48736.476
      582190.057, 48657.628
      581893.321, 48625.161
    BANK POSITIONS: 0.42600, 0.47700
    WATER ELEVATION: 199.3957, 200.6774, 203.5746
    WATER SURFACE EXTENTS:
      582242.56,   48671.16,   582212.81,   48663.49
      582246.10,   48672.07,   582209.73,   48662.70
      582262.79,   48676.37,   582197.27,   48659.49
    PROFILE ID:Big
        VELOCITIES:
          0.43251,   5.29
          0.44147,   11.31
          0.45140,   11.48
          0.46148,   10.50
          0.46968,   4.35
    PROFILE ID:Bigger
        VELOCITIES:
          0.42484,   1.25
          0.43231,   5.93
          0.44145,   12.24
          0.45141,   12.40
          0.46151,   11.44
          0.47008,   4.88
          0.47839,   1.21
    PROFILE ID:Biggest
        VELOCITIES:
          0.41496,   3.61
          0.43201,   6.82
          0.44142,   13.32
          0.45143,   13.44
          0.46155,   12.59
          0.47070,   6.09
          0.48537,   3.64
  END:
```

```
many cross sections omitted

  CROSS-SECTION:
    STREAM ID: Wailupe
    REACH ID: Upper
    STATION: 6682.474
    CUT LINE:
      582641.922, 47366.533
      582444.447, 47335.449
      582336.567, 47337.277
      582062.296, 47381.161
    BANK POSITIONS:0.38000, 0.47199
    WATER ELEVATION: 133.7104, 135.6018, 139.3349
    WATER SURFACE EXTENTS:
      582417.69,   47335.90,   582377.28,   47336.59
      582419.39,   47335.87,   582375.14,   47336.62
      582437.89,   47335.56,   582370.92,   47336.69
    PROFILE ID:Big
        VELOCITIES:
          0.39480,   4.79
          0.40794,   10.33
          0.42583,   11.09
          0.44240,   7.20
          0.45444,   1.62
    PROFILE ID:Bigger
        VELOCITIES:
          0.39368,   4.71
          0.40788,   10.07
          0.42586,   10.67
          0.44287,   7.40
          0.45565,   2.49
    PROFILE ID:Biggest
        VELOCITIES:
          0.36681,   2.52
          0.39108,   5.05
          0.40781,   10.10
          0.42589,   10.52
          0.44336,   7.84
          0.45805,   3.38
  END:

  CROSS-SECTION:
    STREAM ID: Kulai Gorge
    REACH ID: Tributary
    STATION: 1089.584
    CUT LINE:
      583337.968, 47187.952
      583207.93, 47327.062
      583153.496, 47381.496
      583126.279, 47608.306
    BANK POSITIONS: 0.37300, 0.57900
    WATER ELEVATION: 219.1924, 220.2025, 221.5454
    WATER SURFACE EXTENTS:
```

```
      583192.52,   47342.48,   583177.65,   47357.34
      583193.66,   47341.33,   583176.64,   47358.36
      583195.18,   47339.81,   583175.29,   47359.70
    PROFILE ID: Big
        VELOCITIES:
          0.44533,   10.34
          0.46033,   8.44
    PROFILE ID: Bigger
        VELOCITIES:
          0.44432,   11.33
          0.46129,   9.32
    PROFILE ID: Biggest
        VELOCITIES:
          0.44296,   12.44
          0.46257,   10.34
  END:
```

many cross sections omitted

```
  CROSS-SECTION:
    STREAM ID: Kulai Gorge
    REACH ID: Tributary
    STATION: 273.138
    CUT LINE:
      582546.842, 47088.605
      582555.984, 47189.171
      582550.499, 47240.368
      582552.327, 47295.223
    BANK POSITIONS: 0.54099, 0.65500
    WATER ELEVATION: 135.2666, 136.3284, 137.8818
    WATER SURFACE EXTENTS:
      582554.32,   47204.74,   582552.81,   47218.77
      582554.37,   47204.24,   582552.76,   47219.30
      582554.45,   47203.51,   582552.67,   47220.11
    PROFILE ID: Big
        VELOCITIES:
          0.56337,   1.54
          0.57644,   10.95
          0.59787,   15.34
          0.61730,   13.10
    PROFILE ID:  Bigger
        VELOCITIES:
          0.56256,   2.97
          0.57620,   12.59
          0.59789,   17.18
          0.61875,   14.67
          0.63251,   1.86
    PROFILE ID: Biggest
        VELOCITIES:
          0.56137,   4.30
          0.57597,   14.31
          0.59791,   19.12
          0.61927,   16.89
```

```
        0.63330,   5.09
END:

CROSS-SECTION:
  STREAM ID: Wailupe
  REACH ID: Lower
  STATION: 6269.258
  CUT LINE:
    582723.174, 46846.449
    582426.438, 46878.916
    581953.514, 47082.992
  BANK POSITIONS: 0.52401, 0.56900
  WATER ELEVATION: 123.9078, 125.2825, 127.845
  WATER SURFACE EXTENTS:
    582309.53,   46929.36,   582276.02,   46943.82
    582309.55,   46929.35,   582275.92,   46943.87
    582326.40,   46922.09,   582195.49,   46978.57
  PROFILE ID: Big
      VELOCITIES:
        0.52365,   2.47
        0.52857,   9.22
        0.53762,   10.37
        0.54643,   10.39
        0.55542,   10.13
        0.56408,   1.21
  PROFILE ID: Bigger
      VELOCITIES:
        0.52363,   2.70
        0.52856,   10.14
        0.53761,   11.38
        0.54644,   11.40
        0.55543,   11.15
        0.56380,   1.27
  PROFILE ID: Biggest
      VELOCITIES:
        0.51786,   2.90
        0.52855,   12.07
        0.53759,   13.52
        0.54645,   13.53
        0.55544,   13.29
        0.56421,   1.42
        0.60838,   1.12
        0.66018,   0.35
END:

Many cross sections (and interpolated cross sections) omitted

CROSS-SECTION:
  STREAM ID: Wailupe
  REACH ID: Lower
  STATION: 220.827
  CUT LINE:
    586214.122, 42127.918
```

```
      581980.991, 40806.059
    BANK POSITIONS: 0.50300, 0.51500
    LEVEE POSITIONS:
      1, 0.345, 7.123
    INEFFECTIVE POSITIONS:
      1, 0.000, 0.412,5.987
    WATER ELEVATION: 5.503006, 5.881266, 6.600093
    WATER SURFACE EXTENTS:
      586214.12,   42127.92,   583114.30,   41159.95
      586214.12,   42127.92,   583049.46,   41139.70
      586214.12,   42127.92,   581980.99,   40806.06
    PROFILE ID: Big
        VELOCITIES:
          0.00851,   0.55
         many lines omitted
          0.71676,   0.18
    PROFILE ID: Bigger
        VELOCITIES:
          0.00855,   0.63
         many lines omitted
          0.72186,   0.25
    PROFILE ID: Biggest
        VELOCITIES:
          0.01783,   0.55
         many lines omitted
          0.99398,   0.11
  END:

END CROSS-SECTIONS:

BEGIN BOUNDS:

  PROFILE LIMITS:
    PROFILE ID: Big
    POLYGON:
      581893.32, 48625.16
     many lines omitted
      581908.77, 48563.31
    POLYGON:
      583126.27, 47608.3
     many lines omitted
      583090.99, 47539.75
    POLYGON:
      581953.51, 47082.99
     many lines omitted
      581934.96, 47008.78
  END:

  PROFILE LIMITS:
    PROFILE ID: Bigger
      581893.32, 48625.16
     many lines omitted
      581908.77, 48563.31
```

```
    POLYGON:
      583126.27, 47608.3
     many lines omitted
      583090.99, 47539.75
    POLYGON:
      581953.51, 47082.99
     many lines omitted
      581934.96, 47008.78
  END:

  PROFILE LIMITS:
    PROFILE ID: Biggest
      581893.32, 48625.16
    many lines omitted
      581908.77, 48563.31
    POLYGON:
      583126.27, 47608.3
     many lines omitted
      583090.99, 47539.75
    POLYGON:
      581953.51, 47082.99
     many lines omitted
      581934.96, 47008.78
  END:
END BOUNDS:

BEGIN LEVEES:

  LEVEE ID: 1
  SURFACE LINE:
      584579.800, 41808.166, 7.222
   many lines omitted
      584631.334, 41572.921, 5.922
  END:

END LEVEES:

BEGIN INEFFECTIVE AREAS:

  INEFFECTIVE ID: 1
  POLYGON:
      584422.114, 41883.166
      584452.132, 41787.626
   many lines omitted
      584552.852, 41577.321
  END:

END INEFFECTIVE AREAS:

BEGIN STORAGE AREAS:

  SA ID: 1
  POLYGON:
      581919.014, 43565.358
```

```
   many lines omitted
      581895.224, 43443.864
  END:

  ELEVATION-VOLUME:
      20.000, 0
      22.000, 675000
   many lines omitted
      30.000, 421300
  END:

  TERRAIN:
      581898.124, 43566.478, 33.442
      582361.222, 43216.332, 20.369
   many lines omitted
      581867.484, 43432.612, 33.356
  END:

END STORAGE AREAS:
```

Appendix C

HEC-RAS Output Variables

Variable Name	Units	Description
# Barrels	#	Number of barrels in a culvert.
Alpha	-	Alpha - energy weighting coefficient.
Area	sq ft	Flow area of the entire cross section including ineffective flow.
Area Channel	sq ft	Flow area of the main channel including ineffective flow.
Area Left	sq ft	Flow area of the left overbank including ineffective flow.
Area Right	sq ft	Flow area of the right overbank including ineffective flow.
Base WS	ft	Water surface for first profile (used in comparison to encroachment profiles).
Beta	-	Beta - momentum weighting coefficient.
BR Open Area	sq ft	Total area of the entire bridge opening.
BR Open Vel	ft/s	Average velocity inside the bridge opening (Maximum of BU and BD).
Br Sel Mthd	-	Selected bridge hydraulic modeling method.
Breach CL	ft	Center line of weir breach.
Breach WD	ft	Bottom width of weir breach.
Breach Bottom El	ft	Bottom Elevation of weir breach.
Breach Top El	ft	Top Elevation of weir breach.
Breach SSL	ft	Left side slope of weir breach.
Breach SSR	ft	Right side slope of weir breach.
C & E Loss	ft	Contraction or expansion loss between two cross sections.
Center Station	ft	Stationing of the center of the main channel.
Ch Sta L	ft	Left station of main channel.
Ch Sta R	ft	Right station of main channel.
Clv EG No Wr	ft	Energy grade elevation at the culvert when calculated without the weir.
Coef of Q	-	WSPRO bridge method coefficient of discharge.
Conv. Chnl	cfs	Conveyance of main channel.
Conv. Left	cfs	Conveyance of left overbank.
Conv. Ratio	-	Ratio of the conveyance of the current cross section to the conveyance of the downstream cross section.
Conv. Right	cfs	Conveyance of right overbank.
Conv. Total	cfs	Conveyance of total cross section.
Crit Depth	ft	Critical depth. Corresponds to critical water surface.
Crit E.G.	ft	Critical energy elevation. Minimum energy on the energy versus depth curve.
Crit Enrgy 1	ft	Energy associated with first critical depth.
Crit Enrgy 2	ft	Energy associated with second critical depth.
Crit Enrgy 3	ft	Energy associated with third critical depth.
Crit Num	#	Number of critical depths found.
Crit W.S.	ft	Critical water surface elevation. Water surface corresponding to the minimum energy on the energy versus depth curve.

Crit W.S. 1	ft	Water surface elevation of first critical depth.
Crit W.S. 2	ft	Water surface elevation of second critical depth.
Crit W.S. 3	ft	Water surface elevation of third critical depth.
Culv Crt Depth	ft	Critical depth inside the culvert.
Culv Depth Blocked	ft	Depth of fill in a culvert.
Culv EG In	ft	Energy gradeline inside the culvert at the inlet.
Culv EG Out	ft	Energy gradeline inside the culvert at the outlet.
Culv Ent Lss	ft	Culvert entrance loss (energy loss due only to entrance).
Culv Ext Lss	ft	Culvert exit loss (energy loss due to exit).
Culv Frctn Ls	ft	Friction loss through the culvert barrel.
Culv Ful Lngh	ft	The length that the culvert flows full.
Culv Inlet Mann n		The composite n value at the culvert inlet.
Culv Inv El Dn	ft	Culvert inside invert elevation downstream.
Culv Inv El Up	ft	Culvert inside invert elevation upstream.
Culv Length	ft	Length of the culvert barrel.
Culv Nml Depth	ft	Normal depth for this culvert (and flow).
Culv Outlet Mann n		The composite n value at the culvert outlet.
Culv Q	cfs	Flow through all barrels in a culvert group.
Culv Vel DS	ft/s	Velocity inside of culvert at inlet.
Culv Vel US	ft/s	Velocity inside of culvert at outlet.
Culv WS In	ft	Water surface elevation inside the culvert at the inlet.
Culv WS Out	ft	Water surface elevation inside the culvert at the outlet.
Cum Ch Len	ft	Cumulative Channel Length.
Deck Width	ft	Width of bridge/culvert Deck (top of embankment), in direction of flow.
Delta EG	ft	Change in energy grade line through culvert(s) and bridge(s).
Delta WS	ft	Change in water surface through culvert(s) and bridge(s).
Dist Center L	ft	Distance from center of channel to left encroachment.
Dist Center R	ft	Distance from center of channel to right encroachment.
E.G. DS	ft	Energy grade elevation at downsteam end of bridge or culvert.
E.G. Elev	ft	Energy gradeline for calculated WS Elev.
E.G. IC	ft	Upstream energy gradeline at culvert based on inlet control.
E.G. OC	ft	Upstream energy gradeline at culvert based on outlet control.
E.G. Slope	ft/ft	Slope of the energy grade line.
E.G. US.	ft	Energy grade elevation at upstream end of bridge or culvert (final answer).
Enc Method	-	Encroachment method used at this cross section.
Enc Sta L	ft	Left station of encroachment.
Enc Sta R	ft	Right station of encroachment.
Enc Val 1	ft	Target for encroachment analysis.
Enc Val 2	ft	Second target for encroachment analysis.
Encr WD	ft	Top width between encroachments.
Energy EG	ft	Energy grade elevation upstream of bridge for energy only method.
Energy WS	ft	Water surface elevation upstream of bridge for energy only method.
Energy/Wr EG	ft	Energy grade elevation upstream of bridge for low energy and weir method.
Energy/Wr WS	ft	Water surface elevation upstream of bridge for low flow energy method and weir flow.

Flow Area	sq ft	Total area of cross section active flow.
Flow Area Ch	sq ft	Area of main channel active flow.
Flow Area L	sq ft	Area of left overbank active flow.
Flow Area R	sq ft	Area of right overbank active flow.
Frctn Loss	ft	Friction loss between two cross sections.
Frctn Slope	ft/ft	Representative friction slope between two cross sections.
Frctn Slp Md	-	Friction slope averaging method used.
Froude # Chl	-	Froude number for the main channel.
Froude # XS	-	Froude number for the entire cross section.
Gate #Open	#	The number of gates opened in the current group.
Gate Area	sq ft	The flow area in an opened gate.
Gate Group Q	cfs	Flow through all gate openings in a gate group.
Gate Invert	ft	Gate spillway invert elevation.
Gate Open Ht	ft	Height of gate opening.
Gate Submerg	-	Degree of gate submergence. The ratio of the downstream depth above the gate to the upstream depth above the gate.
Headloss	ft	Total energy loss between two cross sections.
Hydr Depth	ft	Hydraulic depth for cross section (Area/Topwidth of active flow).
Hydr Depth C	ft	Hydraulic depth in channel (channel flow area/topwidth of channel flow).
Hydr Depth L	ft	Hydraulic depth in left overbank (left overbank flow area/topwidth of left overbank flow).
Hydr Depth R	ft	Hydraulic depth for right over bank (right overbank flow area/topwidth of right overbank flow).
Ice Btm Chan	ft	The bottom elevation of ice in the main channel.
Ice Btm LOB	ft	The bottom elevation of ice in the left overbank.
Ice Btm ROB	ft	The bottom elevation of ice in the right overbank.
Ice Err	ft	Convergence error in ice thickness for dynamic ice jam.
Ice Thick Chan	ft	Ice thickness in the main channel.
Ice Thick LOB	ft	Ice thickness in the left overbank.
Ice Thick ROB	ft	Ice thickness in the right overbank.
Ice Top Chan	ft	The top elevation of ice in the main channel.
Ice Top LOB	ft	The top elevation of ice in the left overbank.
Ice Top ROB	ft	The top elevation of ice in the right overbank.
Ice Vol Total	cu ft	Cumulative volume of ice in an ice jam.
Ice Vol. Chan	cu ft	Cumulative volume of ice in the main channel for an ice jam.
Ice Vol. LOB	cu ft	Cumulative volume of ice in the left overbank for an ice jam.
Ice Vol. ROB	cu ft	Cumulative volume of ice in the right overbank for an ice jam.
Ice WS Err	ft	Convergence error in water surface for dynamic ice jam.
Ineff El Left	ft	The elevation of the left ineffective area.
Ineff El Right	ft	The elevation of the right ineffective area.
Inflow	cfs	Net inflow into a storage area.
Invert Slope	ft/ft	The slope from the invert of this cross section to the next cross section downstream.
IW Gate Flow	cfs	Total flow through all of the gate groups of an inline weir/spillway.
K Perc L	ft	Conveyance reduction from left encroachment.
K Perc R	ft	Conveyance reduction from right encroachment.

L. Freeboard	ft	The freeboard in the main channel at the left bank (left bank elevation minus water surface elevation).
L. Levee Frbrd	ft	The freeboard before the left levee is over-topped.
Left Sta Eff	ft	Furthest left station where there is effective flow.
Length Chnl	ft	Downstream reach length of the main channel.
Length Left	ft	Downstream reach length of the left overbank.
Length Rght	ft	Downstream reach length of the right overbank.
Length Wtd.	ft	Weighted cross section reach length, based on flow distribution, in left bank, channel, and right bank.
Levee El Left	ft	The elevation of the left levee.
Levee El Right	ft	The elevation of the right levee.
LOB Elev	ft	The ground elevation at the left bank of the main channel.
Mann Comp	-	Composite Manning's n value for main channel.
Mann Wtd Chnl		Conveyance weighted Manning's n for the main channel.
Mann Wtd Chnl		Conveyance weighted Manning's n for the left overbank.
Mann Wtd Rght		Conveyance weighted Manning's n for the right overbank.
Mann Wtd Total		Manning's n value for the total main cross section.
Max Chl Dpth	ft	Maximum main channel depth.
Min Ch El	ft	Minimum main channel elevation.
Min El	ft	Minimum overall section elevation.
Min El Prs	ft	Elevation at the bridge when pressure flow begins.
Min Error	ft	The minimum error, between the calculated and assumed water surfaces when balancing the energy equation.
Min El Weir Flow	ft	Elevation where weir flow begins.
Min Weir El	ft	Minimum elevation of a weir.
Momen. EG	ft	Energy grade elevation upstream of bridge for momentum method.
Momen. WS	ft	Water surface elevation upstream of bridge for momentum method.
Net Flux	cfs	Net inflow - outflow for a storage area.
Num Trials	#	Current number (or final number) of trials attempted before the energy equation is balanced.
Obs WS	ft	Observed water surface elevation.
Outflow	cfs	Net outflow into a storage area.
Perc Q Leaving		Percentage of flow leaving through a lateral weir.
Piping Flow	ft	Flow from piping weir failure.
Power Chan	lb/ft s	Total stream power in main channel (main channel shear stress times main channel average velocity). Used in Yang's and other sediment transport equations.
Power LOB	lb/ft s	Total stream power in left overbank (left overbank shear stress times left overbank average velocity). Used in Yang's and other sediment transport equations.
Power ROB	lb/ft s	Total stream power in right overbank (right overbank shear stress times right overbank average velocity). Used in Yang's and other sediment transport equations.
Power Total	lb/ft s	Total stream power (total cross section shear stress times total cross section average velocity). Used in Yang's and other sediment transport equations.
Prof Delta EG	ft	Difference in EG between current profile and EG for first profile.

Prof Delta WS	ft	Difference in WS between current profile and WS for first profile.
Profile	#	Profile number.
Prs O EG	ft	Energy grade elevation upstream of bridge for pressure only method.
Prs O WS	ft	Water surface elevation upstream of bridge for pressure only method.
Prs/Wr EG	ft	Energy grade elevation upstream of bridge for pressure and/or weir method.
Prs/Wr WS	ft	Water surface elevation upstream of bridge for pressure and/or weir method.
Pumping Head	ft	Pumping head for the pump station.
Q Barrel	cfs	Flow through one barrel in a culvert group.
Q Bridge	cfs	Flow through the bridge opening.
Q Channel	cfs	Flow in main channel.
Q Culv	cfs	Total flow in all culvert groups.
Q DS	cfs	Flow in cross section downstream of lateral weir.
Q Lat RC	cfs	Lateral rating curve flow.
Q Leaving Total	cfs	Total flow leaving in a lateral weir including all gates.
Q Left	cfs	Flow in left overbank.
Q Perc Chan	ft	Percent of flow in main overbank.
Q Perc L	ft	Percent of flow in left overbank.
Q Perc R	ft	Percent of flow in right overbank.
Q Pump Group	cfs	Pump group flow.
Q Pump Station	cfs	Total flow in all pump groups in a pump station.
Q Right	cfs	Flow in right overbank.
Q Total	cfs	Total flow in cross section.
Q US	cfs	Flow in cross section upstream of a lateral weir.
Q Weir	cfs	Flow over the weir.
R. Freeboard	ft	The freeboard in the main channel at the right bank (right bank elevation minus water surface elevation).
R. Levee Frbrd	ft	The freeboard before the right levee is over-topped.
Rght Sta Eff	ft	Furthest right station that still has effective flow.
ROB Elev	ft	The ground elevation at the right bank of the main channel.
SA Area	acres	Surface area of a storage area.
SA Chan	acres	Cumulative surface area for main channel from the bottom of the reach.
SA Left	acres	Cumulative surface area for left overbank from the bottom of the reach.
SA Min El	ft	Minimum elevation of a storage area.
SA Right	acres	Cumulative surface area for right overbank from the bottom of the reach.
SA Total	acres	Cumulative surface area for entire cross section from the bottom of the reach.
SA Volume	acre-ft	Storage volume of a storage area.
Shear Chan	lb/sq ft	Shear stress in main channel ($\gamma R_{CH} S_f$).
Shear LOB	lb/sq ft	Shear stress in left overbank ($\gamma R_{LOB} S_f$).
Shear ROB	lb/sq ft	Shear stress in right overbank ($\gamma R_{ROB} S_f$).
Shear Total	lb/sq ft	Shear stress in total section ($\gamma R_T S_f$).

Spc Force PR	cu ft	Specific force prime. For mixed flow, the specific force at this cross section for the flow regime that does not control.
Specif Force	cu ft	The specific force for this cross section at the computed water surface elevation. $SF = A_T Y_{cent} + (Q^2)/(gA_{act})$
Sta W.S. Lft	ft	Left station where water intersects the ground.
Sta W.S. Rgt	ft	Right station where water intersects the ground.
Std Stp Case	#	Standard step method used to determine WSEL (1 = successful convergence, 2 = minimum error, 3 = resorted to critical depth).
Top W Act Chan	ft	Top width of the wetted channel, not including ineffective flow.
Top W Act Left	ft	Top width of the wetted left bank, not including ineffective flow.
Top W Act Right	ft	Top width of the wetted right bank, not including ineffective flow.
Top W Chnl	ft	Top width of the main channel. Does not include 'islands', but it does include ineffective flow.
Top W Left	ft	Top width of the left overbank. Does not include 'islands', but it does include ineffective flow.
Top W Right	ft	Top width of the right overbank. Does not include 'islands', but it does include ineffective flow.
Top Wdth Act	ft	Top width of the wetted cross section, not including ineffective flow.
Top Width	ft	Top width of the wetted cross section.
Total Gate Flow	cfs	Total flow through all of the gate groups of an inline/lateral weir.
Trvl Tme Avg	hrs	Cumulative travel time based on the average velocity of the entire cross section, per reach.
Trvl Tme Chl	hrs	Cumulative travel time based on the average velocity of the main channel, per reach.
Vel Chnl	ft/s	Average velocity of flow in main channel.
Vel Head	ft	Velocity head.
Vel Left	ft/s	Average velocity of flow in left overbank.
Vel Right	ft/s	Average velocity of flow in right overbank.
Vel Total	ft/s	Average velocity of flow in total cross section.
Vol Chan	acre-ft	Cumulative volume of water in the channel (including ineffective flow).
Vol Left	acre-ft	Cumulative volume of water in the left overbank (including ineffective flow).
Vol Right	acre-ft	Cumulative volume of water in the right overbank (including ineffective flow).
Volume	acre-ft	Cumulative volume of water in the direction of computations (including ineffective flow).
W.P. Channel	ft	Wetted perimeter of main channel.
W.P. Left	ft	Wetted perimeter of left overbank.
W.P. Right	ft	Wetted perimeter of right overbank.
W.P. Total	ft	Wetted perimeter of total cross section.
W.S. DS	ft	Water surface downstream of a bridge, culvert, or weir.
W.S. Elev	ft	Calculated water surface from energy equation.
WS Inlet	ft	WS at the inlet of a pump station.

WS Outlet	ft	WS at the outlet of a pump station.
W.S. Prime	ft	Water surface prime. For mixed flow, the water surface of the flow regime that does not control.
W.S. US.	ft	Water surface elevation upstream of bridge or culvert.
Weir Avg Depth	ft	The average depth of flow over the weir.
Weir Max Depth	ft	The maximum depth of flow over the weir.
Weir Sta DS	ft	Downstream station where weir flow ends.
Weir Sta Lft	ft	Station where flow starts on the left side of weir.
Weir Sta Rgt	ft	Station where flow ends on the right side of weir.
Weir Sta US	ft	Upstream station for weir flow starts.
Weir Submerg	-	The ratio of the downstream depth above the weir to the upstream depth above the weir.
Wr Flw Area	sq ft	Area of the flow going over the weir.
Wr Top Wdth	ft	Top width of water over the weir.
WS Air Entr.	ft	Water surface elevation accounting for air entrainment.
WSPRO EG	ft	Energy grade elevation upstream of bridge for the WSPRO method.
WSPRO WS	ft	Water surface elevation upstream of bridge for the WSPRO method.
Wtd. n Chnl	-	Conveyance weighted Manning's n for the main channel.
Wtd. n Left	-	Conveyance weighted Manning's n for the left overbank.
Wtd. n Right	-	Conveyance weighted Manning's n for the right overbank.
XS Delta EG	ft	Change in energy gradeline between current section and next one downstream.
XS Delta WS	ft	Change in water surface between current section and next one downstream.
Yarnell EG	ft	Energy grade elevation upstream of bridge for Yarnell method.
Yarnell WS	ft	Water surface elevation upstream of bridge for Yarnell method.