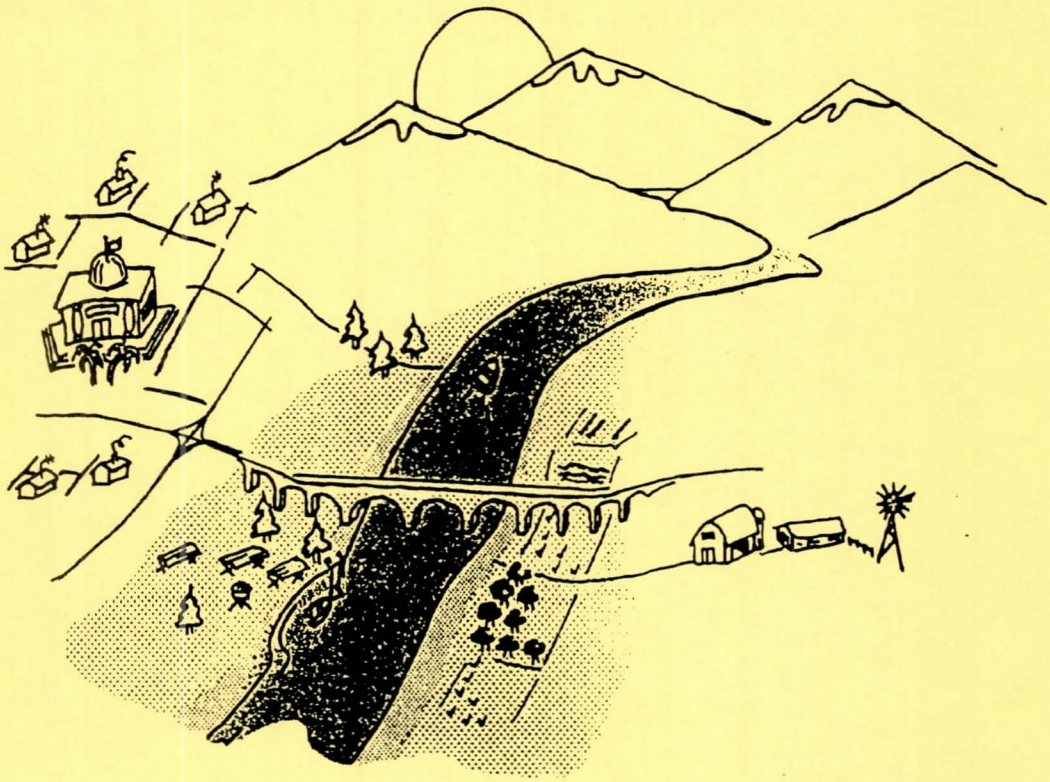
US Army Corps
of Engineers
Hydrologic Engineering Center

Property of
Flood Control District of MC Library
Please Return to
2801 W. Durango
Phoenix, AZ 85009

HEC-RAS
River Analysis System



User's Manual

Version 2.2
September 1998

Approved for Public Release. Distribution Unlimited

CPD-68

HEC-RAS

River Analysis System

User's Manual

Version 2.2

September 1998

US Army Corps of Engineers
Hydrologic Engineering Center
609 Second Street
Davis, CA 95616

(530) 756-1104
(530) 756-8250 FAX

Cover sketch adapted from:

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

Table of Contents

Foreword vi

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 6-3
Editing the Schematic 6-3
Interacting With the Schematic 6-5
Background Pictures 6-7
Cross Section Data 6-7
Entering Cross Section Data 6-7
Editing Cross Section Data 6-10
Cross Section Options 6-11
Plotting Cross Section Data 6-18
Stream Junctions 6-18
Entering Junction Data 6-18
Selecting a Modeling Approach 6-19
Bridges and Culverts 6-19
Cross Section Locations 6-20
Contraction and Expansion Losses 6-22
Bridge Hydraulic Computations 6-23
Entering and Editing Bridge Data 6-25
Bridge Design Editor 6-38
Culvert Hydraulic Computations 6-40
Entering and Editing Culvert Data 6-41
Bridge and Culvert Options 6-46
Bridge and Culvert View Features 6-47
Multiple Bridge and/or Culvert Openings 6-48
Entering Multiple Opening Data 6-49
Defining The Openings 6-52
Multiple Opening Calculations 6-53
Inline Weirs and Gated Spillways 6-54
Entering and Editing Inline Weir and Gated Spillway Data 6-54

Cross Section Interpolation 6-62
River Ice 6-68
Entering and Editing Ice Data 6-68
Entering Ice Data at a Cross Section 6-68
Entering Ice Data Through a Table 6-71
Entering Ice Data at Bridges 6-72
Setting Tolerances for the Ice Jam Calculations 6-73
Viewing and Editing Data Through Tables 6-74
Manning's n or k values 6-74
Reach Lengths 6-76
Contraction and Expansion Coefficients 6-76
River Stationing 6-77
Importing Geometric Data 6-78
GIS Format 6-79
USACE Survey Data Format 6-79
HEC-2 Data Format 6-79
HEC-RAS Data Format 6-80
Graphical Cross Section Editor 6-80
Attaching and Viewing Pictures 6-81
Saving the Geometric Data 6-83

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 7-6
Importing Data from the HEC Data Storage System (HEC-DSS) 7-7
Performing Steady Flow Calculations 7-10
Defining a Plan 7-10
Saving the Plan Information 7-11
Simulation Options 7-11
Starting the Computations 7-16

Chapter 8 Viewing Results 8-1
Cross Sections, Profiles, and Rating Curves 8-1
Viewing Graphics on the Screen 8-1
Graphical Plot Options 8-4
Plotting Velocity Distribution Output 8-6
Plotting Other Variables in Profile 8-8
Plotting One Variable Versus Another 8-8
Sending Graphics to the Printer or Plotter 8-9
Sending Graphics to the Windows Clipboard 8-10
X-Y-Z Perspective Plots 8-10
Tabular Output 8-12
Cross Section Tables 8-12
Cross Section Table Options 8-15
Profile Tables 8-16

User Defined Output Tables 8-18
Sending Tables to the Printer 8-20
Sending Tables to the Windows Clipboard 8-21
Viewing Results From the River System Schematic 8-21
Viewing Ice Information 8-22
Viewing Graphical Ice Information on the Screen 8-22
Viewing Tabular Ice Information 8-25

Chapter 9 Performing a Floodway Encroachment Analysis 9-1
General 9-2
Entering Floodway Encroachment Data 9-3
Performing the Floodway Encroachment Analysis 9-6
Viewing the Floodway Encroachment Results 9-7

Chapter 10 Troubleshooting With HEC-RAS 10-1
Built in Data Checking 10-1
Checking The Data as it is Entered 10-1
Checking Data Before Computations are Performed 10-2
Errors, Warnings, and Notes 10-3
Log Output 10-5
Setting Log File Output 10-5
Viewing The Log File 10-7
Reviewing and Debugging the Normal Output 10-7
Viewing Graphics 10-7
Viewing Tabular Output 10-7
The Occurrence of Critical Depth 10-8
Computational Program Does Not Run To Completion 10-8

Chapter 11 Computing Scour at Bridges 11-1
General Modeling Guidelines 11-2
Entering Bridge Scour Data 11-3
Entering Contraction Scour Data 11-4
Entering Pier Scour Data 11-6
Entering Abutment Scour Data 11-10
Computing Total Bridge Scour 11-13

Chapter 12 Performing Channel Modifications 12-1
General Modeling Guidelines 12-2
Entering Channel Modification Data 12-2
Performing the Channel Modifications 12-6
Comparing Existing and Modified Conditions 12-8

Chapter 13 Using GIS Data With HEC-RAS 13-1
General Modeling Guidelines 13-2
Importing GIS/CADD Data Into HEC-RAS 13-4
Completing The Data and Performing The Computations 13-6
Completing The Geometric Data 13-6
Importing Additional Geometry From The GIS 13-6

Entering Additional Cross Section Data 13-7
Performing The Computations and Viewing Results 13-8
Exporting Computed Results to The GIS or CADD 13-9

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

Foreword

The HEC-RAS software was developed at the Hydrologic Engineering Center (HEC). The software was designed by Mr. Gary W. Brunner, leader of the HEC-RAS development team. The user interface and graphics were programmed by Mr. Mark R. Jensen. The steady flow water surface profiles module was programmed by Mr. Steven S. Piper. The routines that import HEC-2 data were developed by Ms. Joan Klipsch. The cross section interpolation routines were developed by Mr. Alfredo Montalvo. The routines for modeling ice cover and wide river ice jams were developed by Mr. Steven F. Daly of the Cold Regions Research and Engineering Laboratory (CRREL). The Example Applications Manual was put together by Mr. John W. Warner and Mr. Gary W. Brunner.

Many of the HEC staff made contributions in the development of this software, including: Vern R. Bonner, Richard Hayes, John Peters, and Michael Gee. Mr. Darryl Davis was the director during the development of this software.

This manual was written by Mr. Gary W. Brunner.

CHAPTER 1

Introduction

Welcome to the Hydrologic Engineering Center's River Analysis System (HEC-RAS). This software allows you to perform one-dimensional steady flow, unsteady flow, and sediment transport calculations (The current version of HEC-RAS can only perform steady flow calculations. Unsteady flow and sediment transport will be added in future versions).

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 you 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, 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 only supports Steady 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; and multiple bridge and/or culvert opening analysis.

Unsteady Flow Simulation. This component of the HEC-RAS modeling system will be capable of simulating one-dimensional unsteady flow through a full network of open channels. The unsteady flow equation solver will be adapted from Dr. Robert L. Barkau's UNET model (Barkau, 1992 and HEC, 1993). This unsteady flow component was developed primarily for subcritical flow regime calculations.

The hydraulic calculations for cross-sections, bridges, culverts, and other hydraulic structures that were developed for the steady flow component will be incorporated into the unsteady flow module. Additionally, the unsteady flow component will have the ability to model storage areas, navigation dams, tunnels, pumping stations, and levee failures.

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). 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 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 6-7 explain in detail how to enter and edit data, and how to perform the different types of analyses that are available.
- Chapter 8 provides detailed discussions on how to view graphical and tabular output, as well as how to develop user defined tables.
- Chapter 9 describes how to perform a floodway encroachment analysis.
- Chapter 10 provides discussions on "Trouble Shooting" and understanding the most common Errors, Warnings, and Notes.
- Chapter 11 describes how to perform bridge scour computations from within HEC-RAS.
- Chapter 12 describes how to perform channel modifications within HEC-RAS.
- Chapter 13 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.
- 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 disks 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:

- Any IBM or compatible machine with an 80486 processor or higher (a Pentium or higher is recommended).
- A hard disk with at least 10 megabytes of **free** space (20 megabytes or more is recommended).
- A 3 ½" floppy drive.
- A minimum of 16 megabytes of RAM if using Windows 95 or 24 megabytes if using Windows NT (32 or more is recommended).
- A mouse.
- Color VGA or better Video Display (Recommend running in Super VGA (800x600) or higher, and as large a monitor as possible).
- MS Windows 95 or Windows NT 4.0 (or later versions).

Installation Procedure

Installation of the HEC-RAS software is accomplished through the use of the Setup program. When you run the Setup program, you will be asked to set a path for the program and data files. A suggested directory of "\HEC\RAS" will be provided. You may choose to use this directory name or provide one of your own.

▸ **To install the software onto your hard disk from Windows 95 or Windows NT 4.0, do the following:**

1. Insert Disk 1 into the A drive (or B if necessary).
2. Press the **Start** button in the lower left corner of the screen, then select the **Run** option from the menu.
3. Type **a:setup** (or **b:setup** if disk 1 is in the B drive) and press **ENTER**.

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 \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.

This Page Intentionally Left Blank

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 Flow water surface profile calculations, and will include Unsteady Flow, 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:

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).

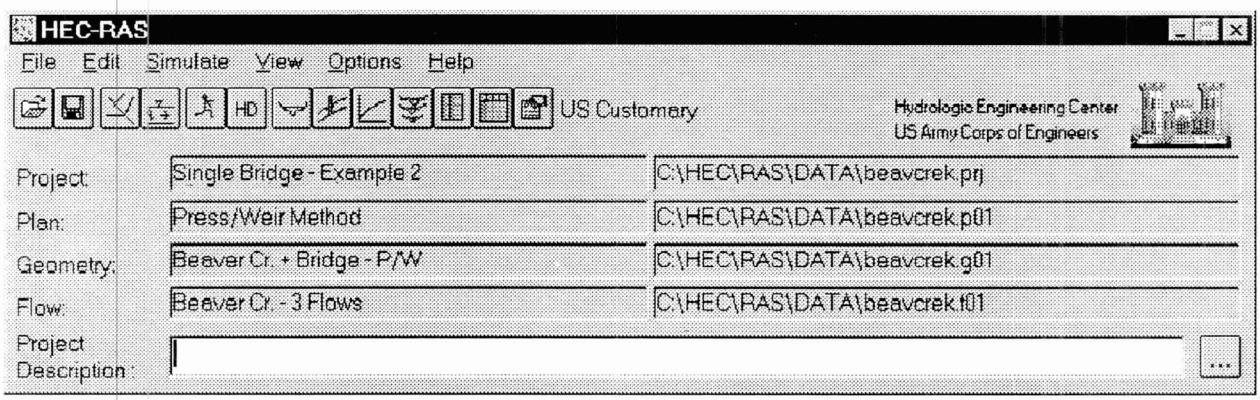


Figure 3.2 The HEC-RAS Main Window

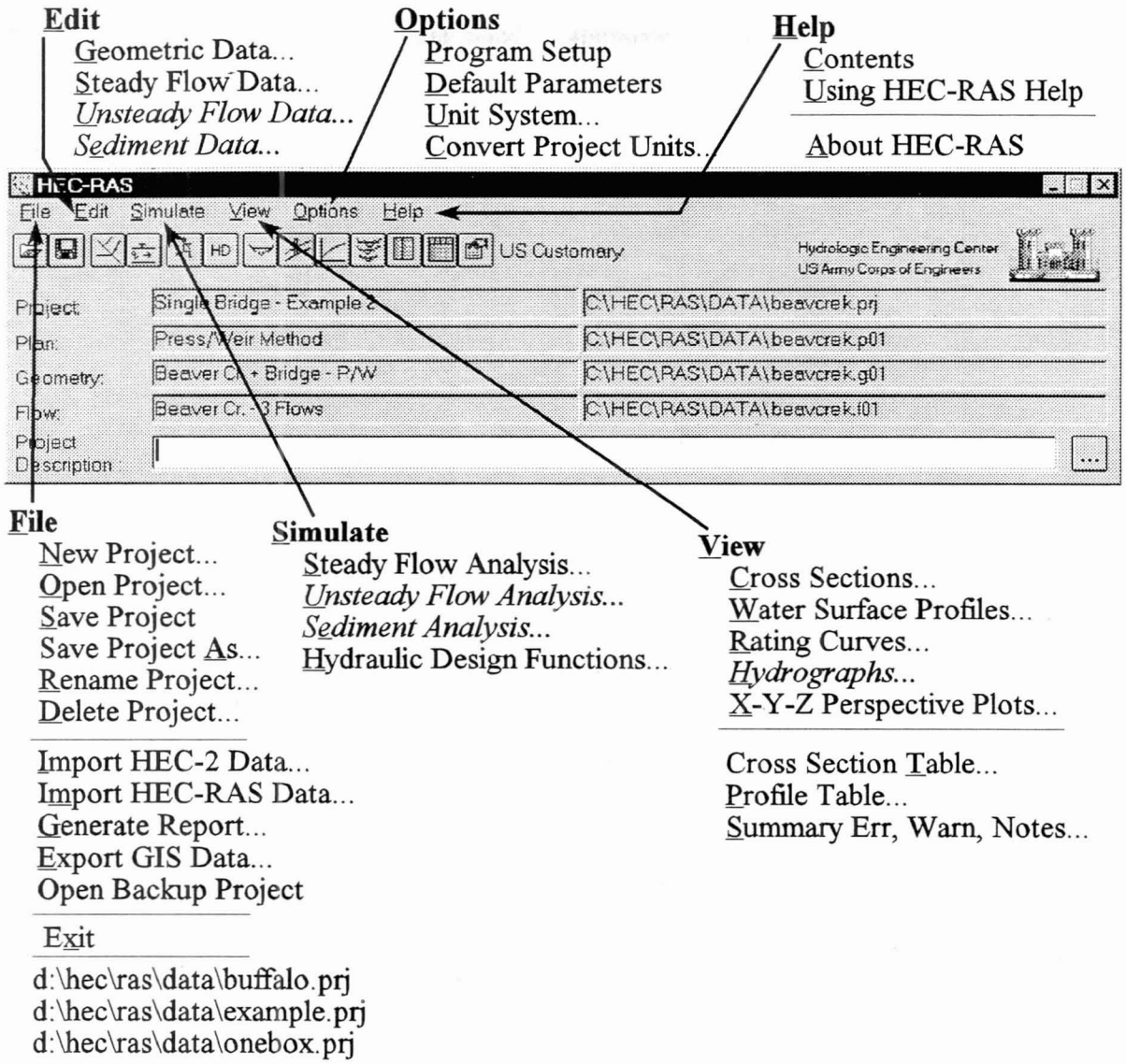


Figure 3.3 HEC-RAS Main Window Menu Bar Structure

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; Import HEC-2 Data; Import HEC-RAS data; Generate Report; Export GIS Data; Open Backup Project; and Exit. In addition, the four 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, only Geometric Data and Steady Flow Data are active.

Simulate: 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, Unsteady Flow Analysis and Sediment Analysis are 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; Rating Curves; X-Y-Z Perspective Plots; Cross Section Tables; Profile Tables; and Summary Err, Warn, Notes. Hydrograph plots are not available yet, but will be included when unsteady flow capabilities are added to the system.

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.

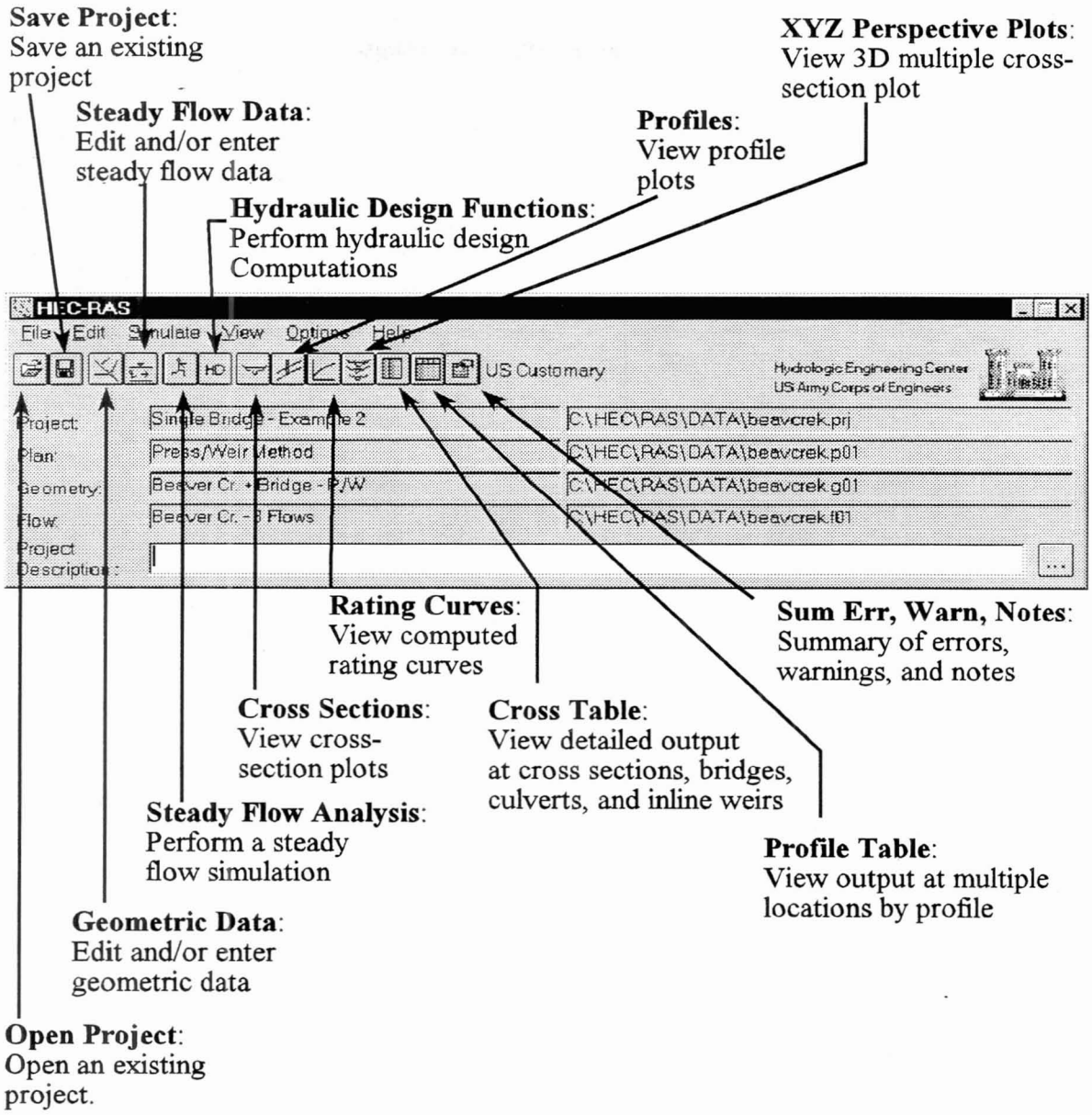


Figure 3.4 HEC-RAS Main Window Button Bar

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.

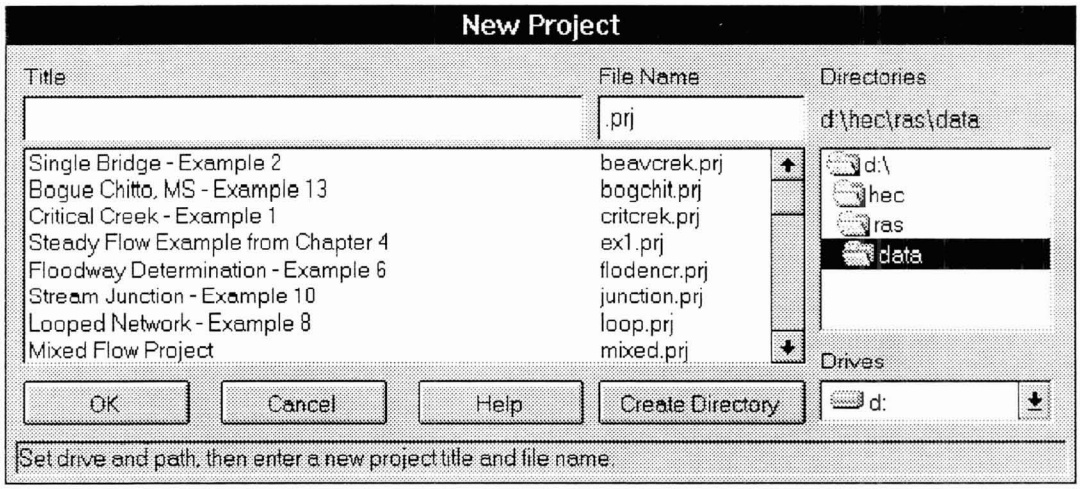


Figure 3.5 New Project window

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), then 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 is accomplished by selecting **Unit System** from the **Options** menu on 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).

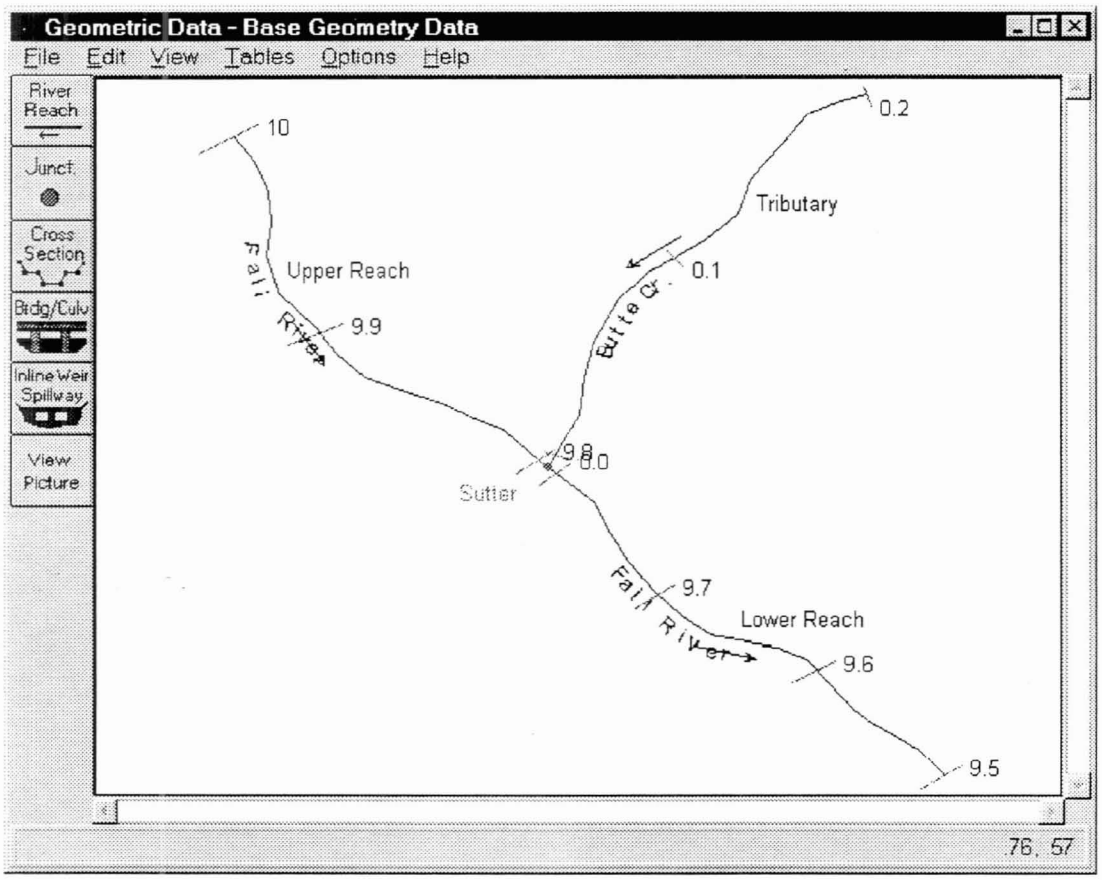


Figure 3.6 Geometric Data Window

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 too 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.**

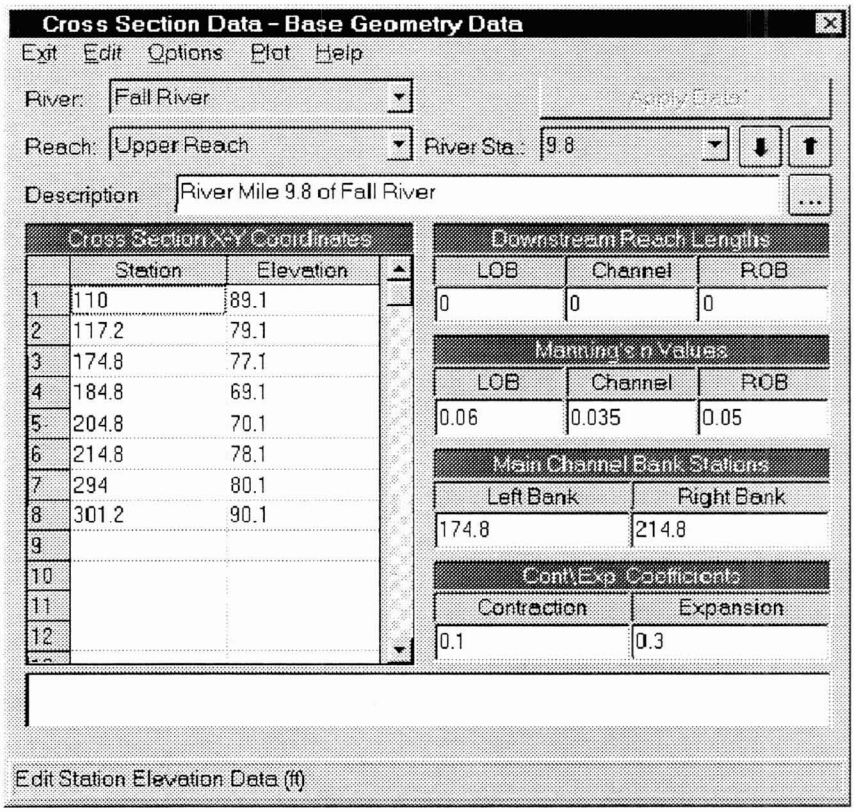


Figure 3.7 Cross Section Data editor

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; ineffective flow areas; levees; blocked obstructions; adding a lid to a cross section; horizontal variation of n or k-values; and setting the maximum number of station and elevation points.

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

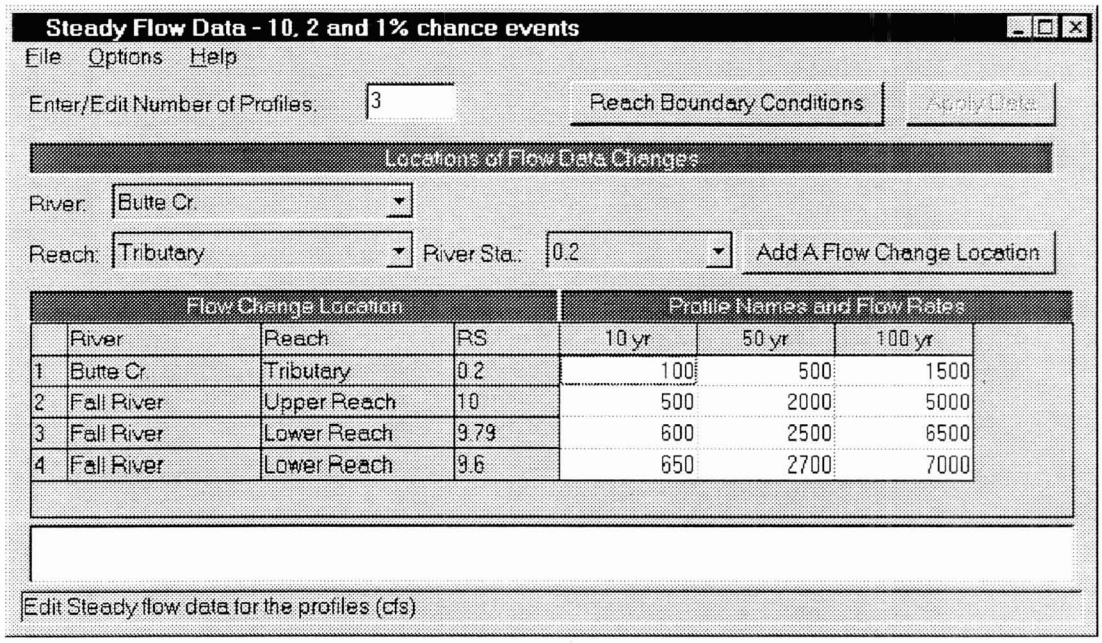
Once the geometric data are entered, the modeler can then enter any flow data that are required. The data entry form for flow data is available under the **Edit** menu bar option on the HEC-RAS main window.

An example of the flow data entry form is shown in Figure 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 **Enter**

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 data, the filename is automatically assigned.



	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 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 two types of calculations that can be performed in the current version of HEC-RAS: Steady Flow Analysis, and Hydraulic Design Functions. The modeler can select any of the available hydraulic analyses from the **Simulate** menu bar option on the HEC-RAS main window. An example of the simulation window is shown in Figure 3.9, which is the Steady Flow Analysis window.

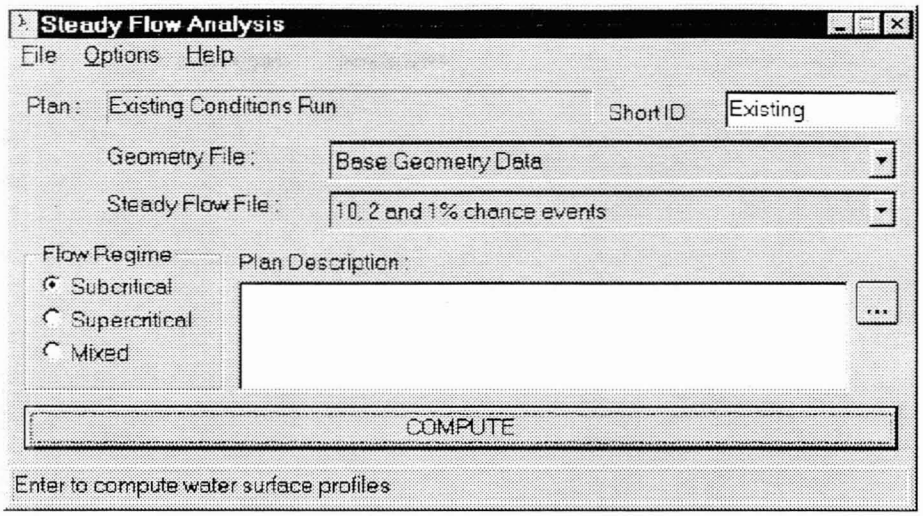


Figure 3.9 Steady Flow Analysis window

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; setting output options; conveyance calculation options; friction slope methods; calculation tolerances; critical depth computation method; 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; tabular output at specific locations (Cross Section Table); tabular output for many locations (Profile Table); 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; selecting which plans, profiles and variables to plot; 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.

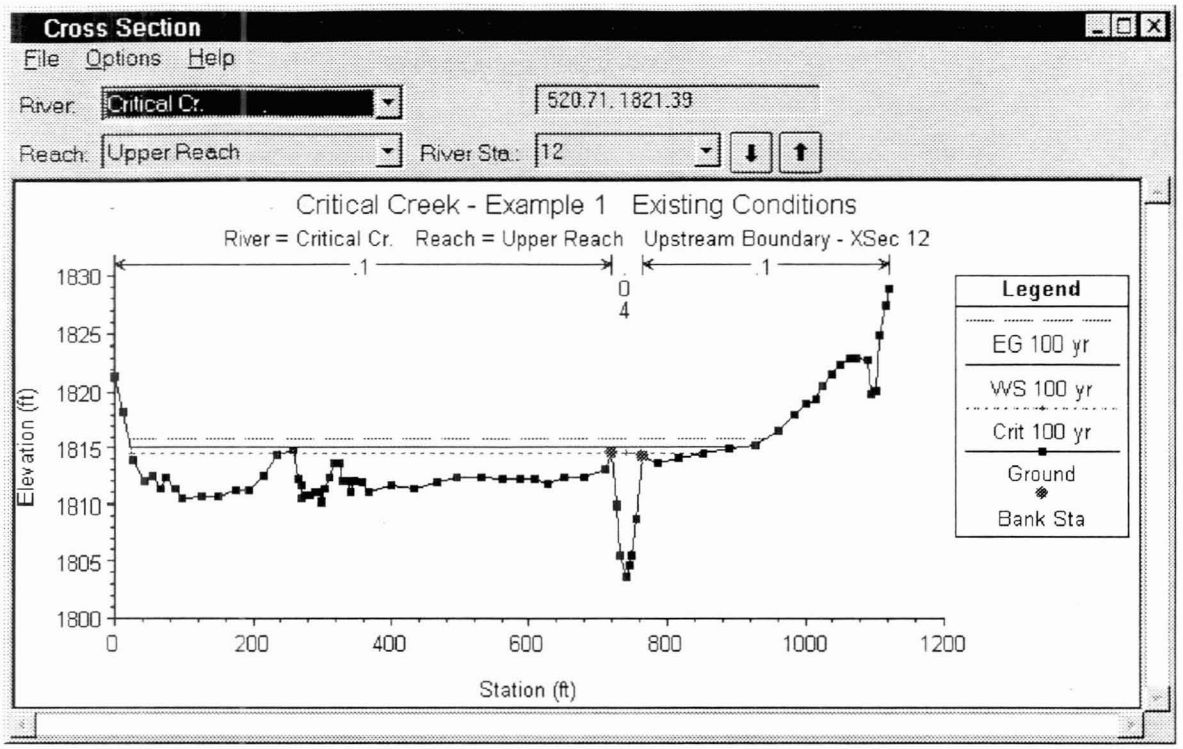


Figure 3.10 Cross Section Plot

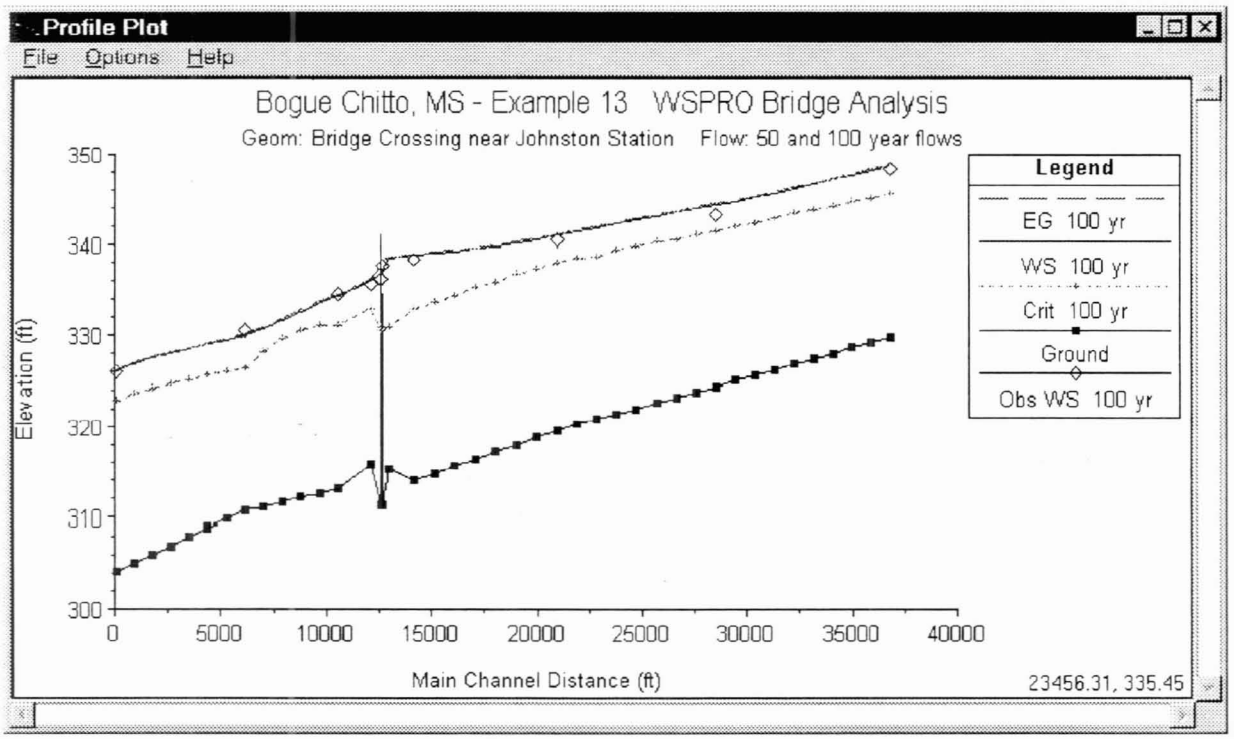


Figure 3.11 Profile Plot

Profile Output Table - Standard Table 1

File Options Std. Tables User Tables Help

HEC-RAS Plan: Modified Geo River: Critical Cr Reach: Upper Reach

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)
Upper Reach	12	9000.00	1803.60	1815.54	1814.46	1816.02	0.004559	8.95	2995.71	915.36
Upper Reach	11	9000.00	1800.70	1810.67	1810.42	1811.90	0.007044	11.20	1888.36	651.40
Upper Reach	10	9000.00	1794.40	1804.63	1803.69	1805.08	0.008683	9.84	2636.20	960.38
Upper Reach	9	9000.00	1788.70	1799.04	1799.31	1800.25	0.012421	13.21	2383.45	1209.84
Upper Reach	8	9500.00	1784.30	1794.30	1793.92	1795.14	0.006072	10.79	3002.35	1193.87
Upper Reach	7	9500.00	1777.20	1789.68	1788.91	1790.95	0.008486	13.88	2049.04	522.90
Upper Reach	6	9500.00	1774.50	1784.29	1784.29	1786.35	0.011147	13.38	1266.10	332.35
Upper Reach	5	9500.00	1768.50	1777.28	1776.80	1778.27	0.009072	11.79	2104.97	596.10
Upper Reach	4	9500.00	1763.00	1772.79	1772.23	1773.49	0.007994	11.30	2501.88	734.37
Upper Reach	3	9500.00	1759.40	1767.85	1765.80	1769.14	0.011972	13.33	1961.94	653.56
Upper Reach	2	9500.00	1753.60	1761.54	1760.03	1762.10	0.009423	10.36	2322.79	682.68
Upper Reach	1	9500.00	1747.40	1756.71	1755.60	1757.21	0.010001	9.91	2404.08	728.01

Total flow in cross section.

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:

- Split flow optimization (SF, etc.)
- Ice (IC)
- Vertical variation of Manning's n values (NV)
- Compute Manning's n from high water marks (J1)
- Internal Rating Curves (RC)
- 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.

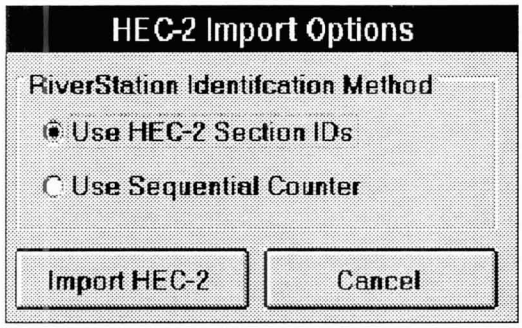


Figure 3.15 Method for Identifying River Stations from HEC-2 Data

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. 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.

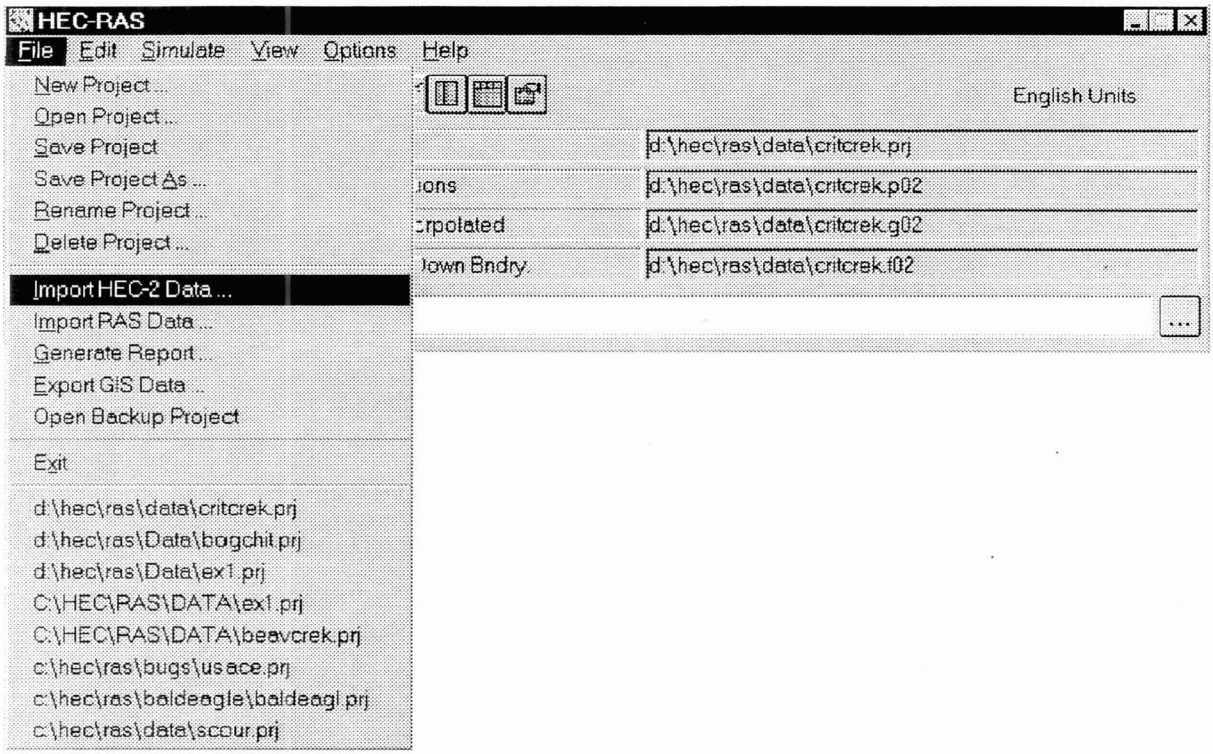


Figure 3.16 HEC-RAS Main Window With File Menu Options Shown

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.

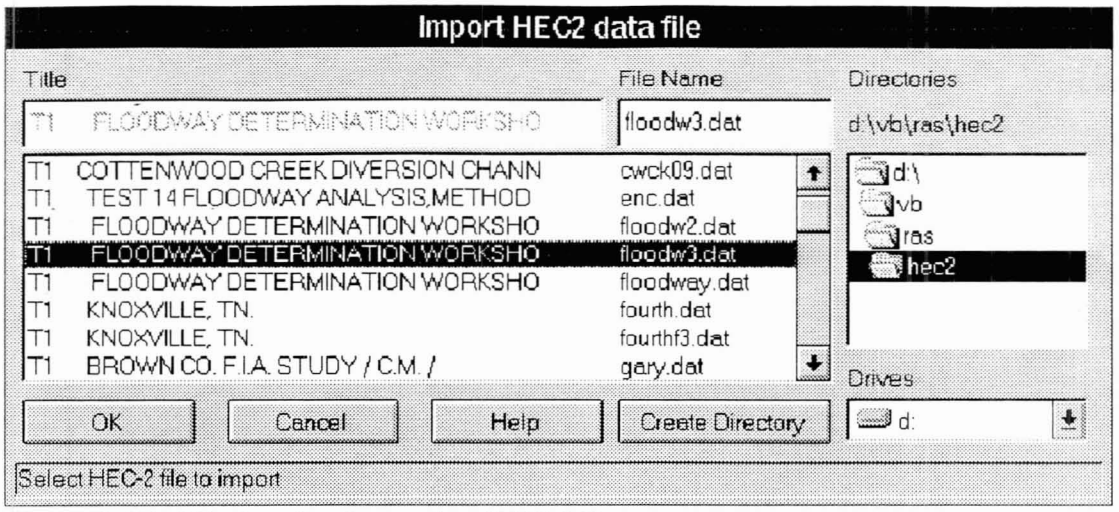


Figure 3.17 Window for Importing HEC-2 Data

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.

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 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.

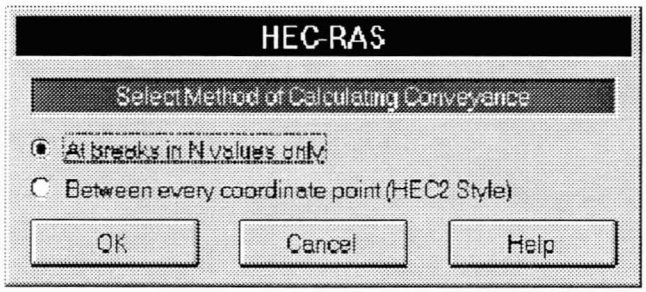


Figure 3.18 HEC-RAS Conveyance Calculation Methods

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).

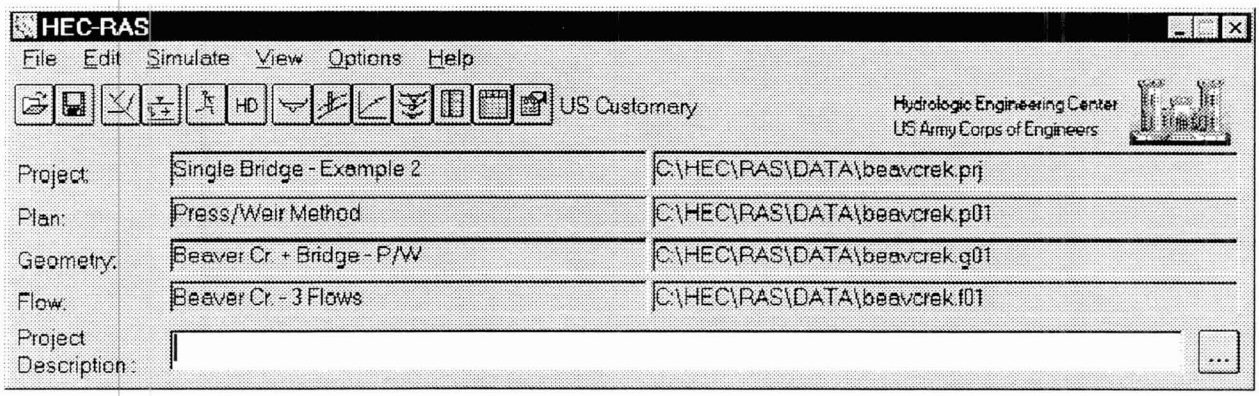


Figure 4.1 HEC-RAS Main Window

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).

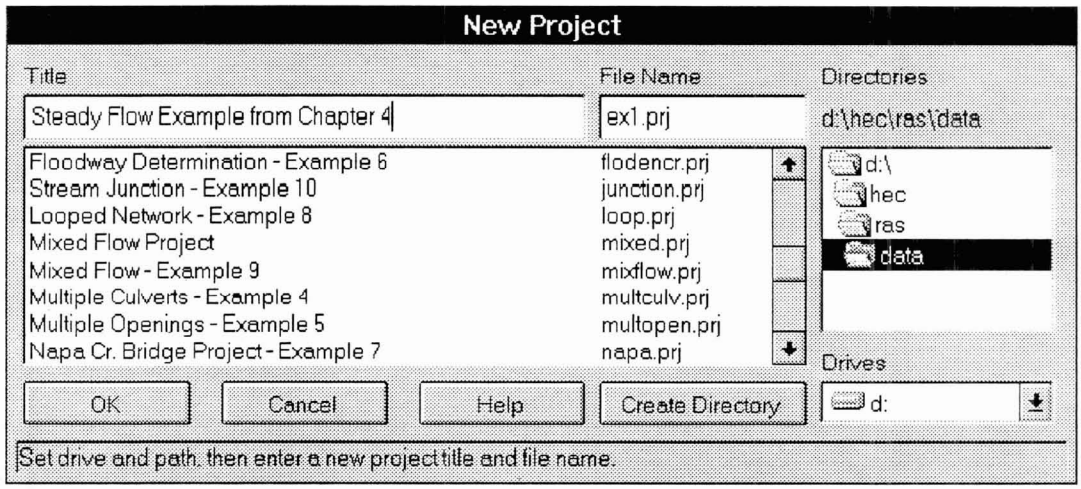


Figure 4.2 New Project Window

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 section data.**

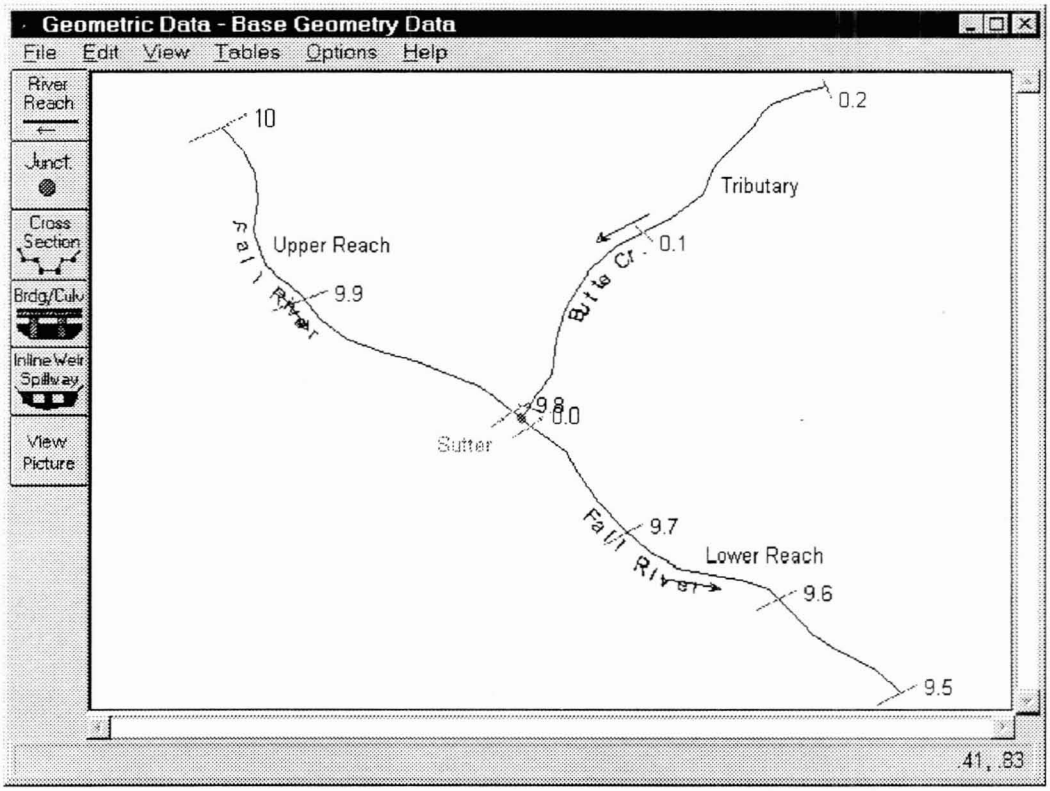


Figure 4.3 Geometric Data Window with example river schematic

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.

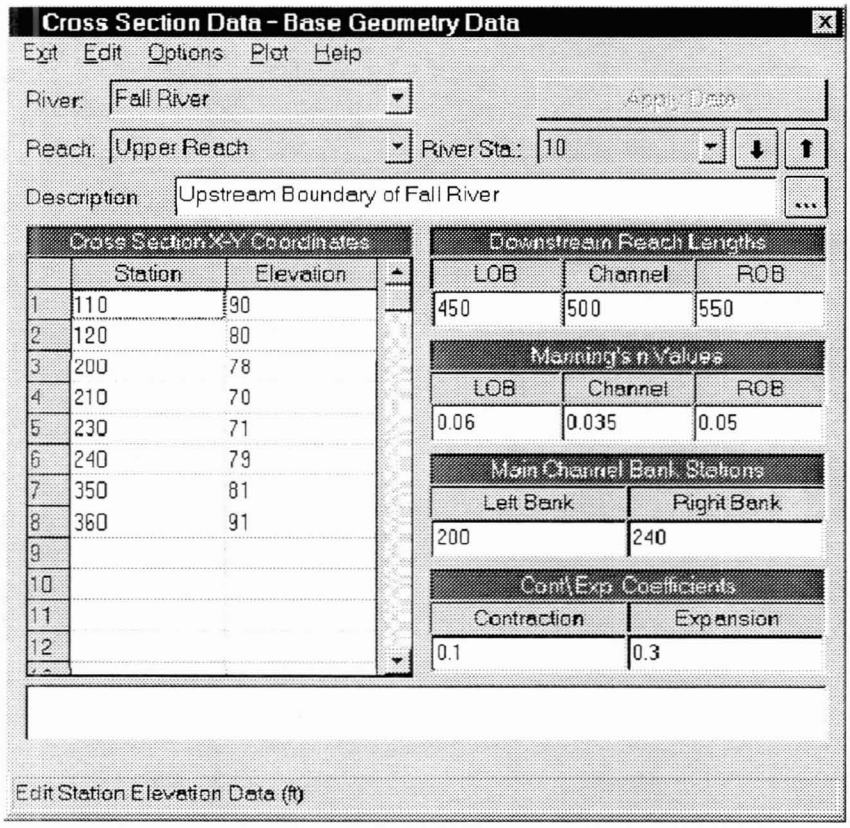


Figure 4.4 Cross Section Data Editor with example data

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.

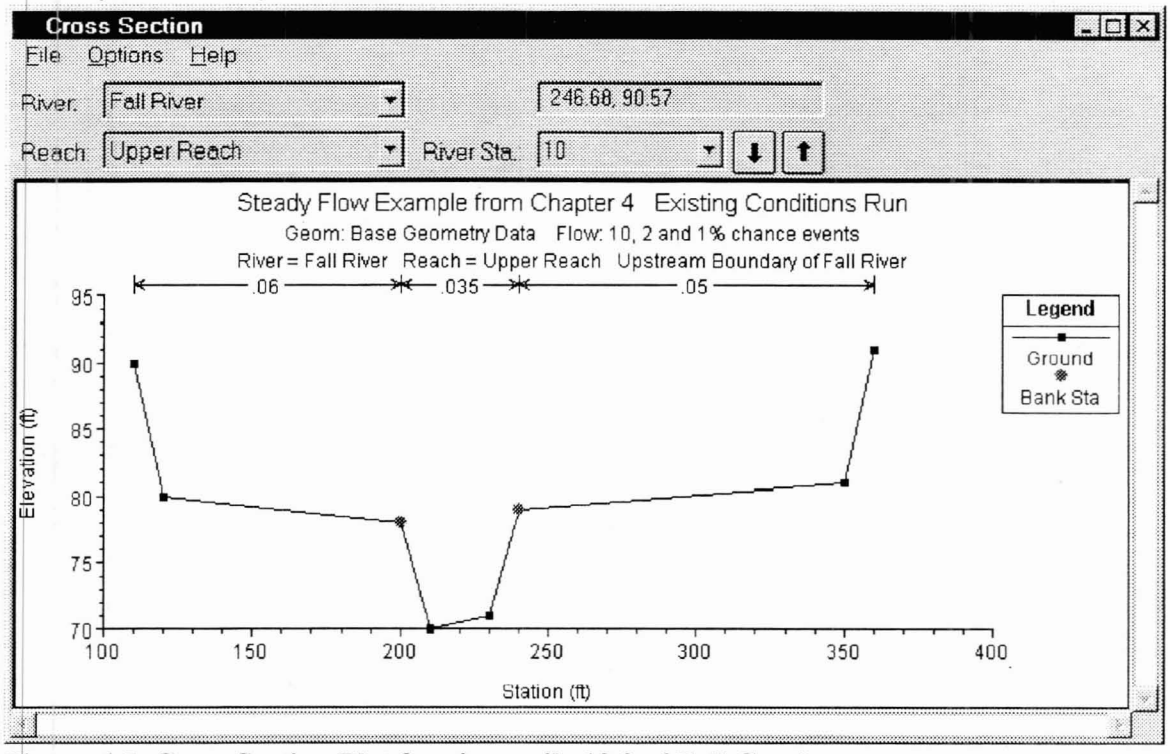


Figure 4.5 Cross Section Plot for river mile 10.0 of Fall Creek

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 description for the cross section 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.

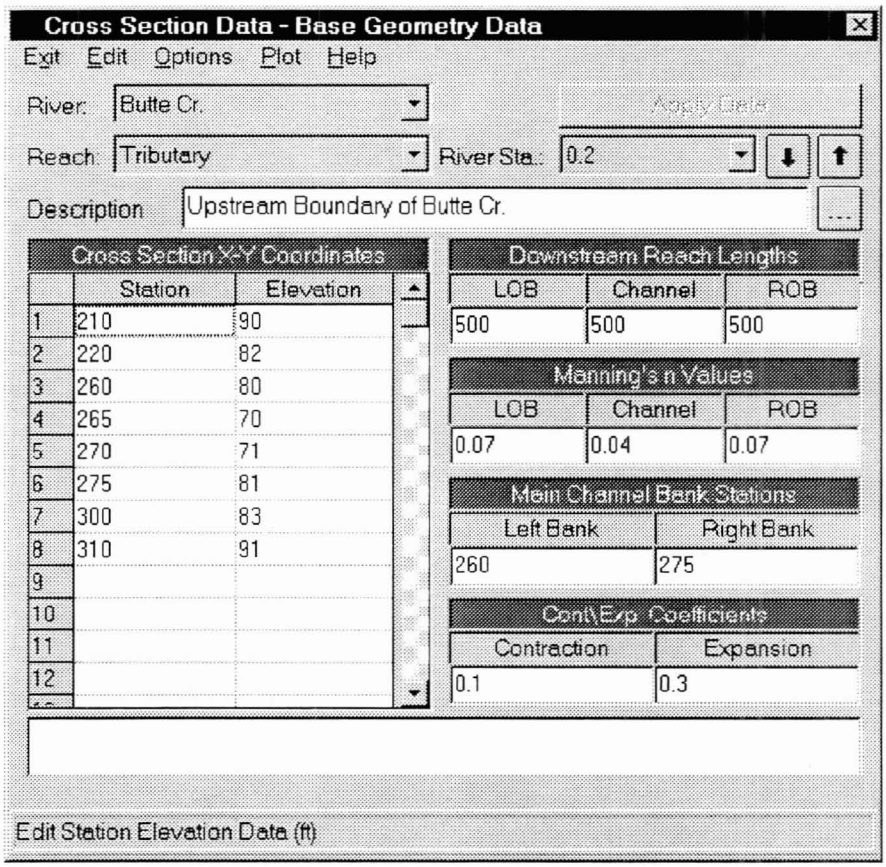


Figure 4.6 Cross Section Editor with river mile 0.2 of Butte Creek

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**. Junction data is entered by pressing the **Junction** button on the Geometric Data window. Enter the junction data as shown in Figure 4.7.

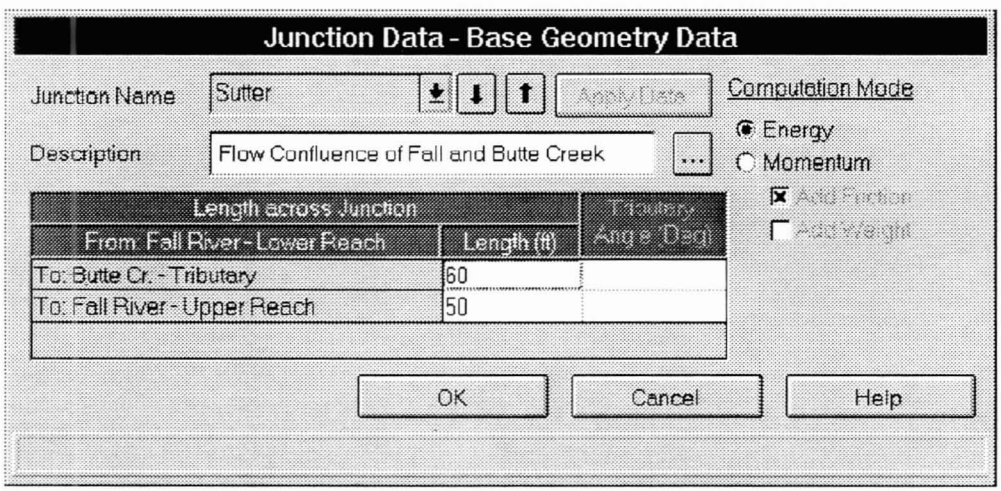


Figure 4.7 Junction Data Editor, with Sutter junction data

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.

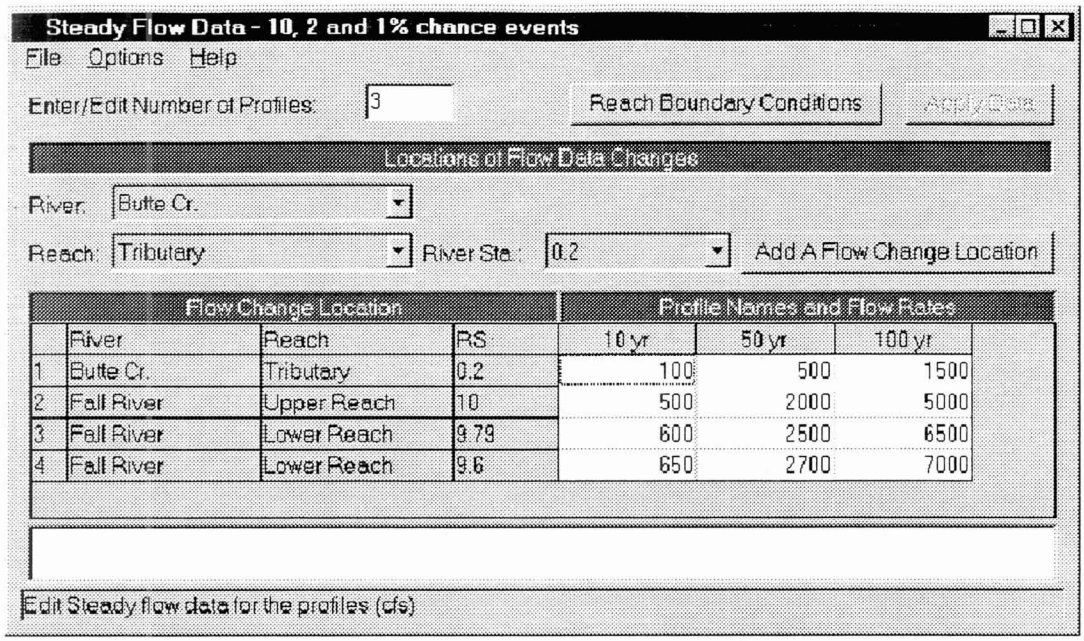
Saving the Geometry Data

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.

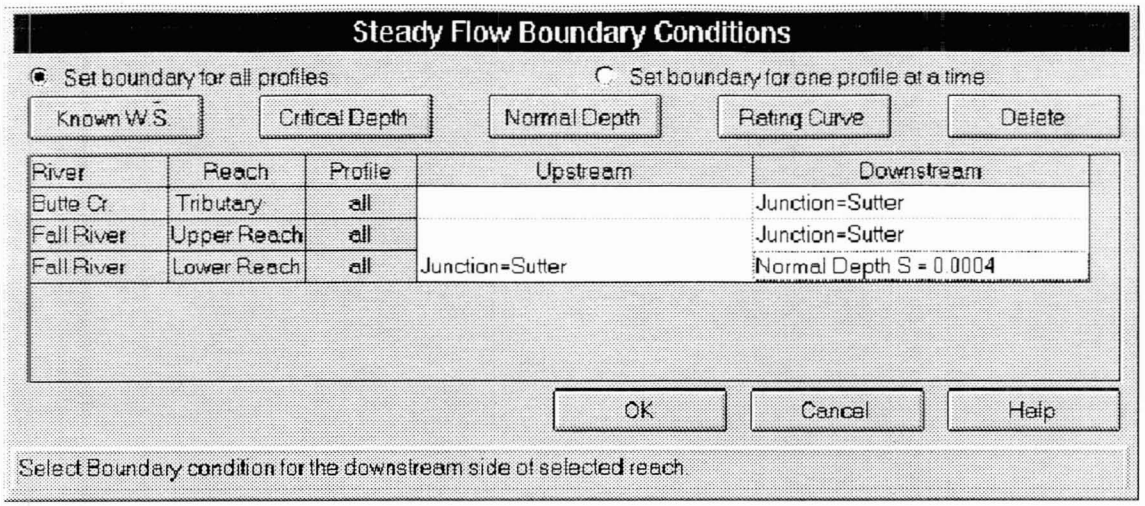


Figure 4.9 Steady Flow Boundary Conditions

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 **Simulate** menu. The Steady Flow Analysis window should appear as shown in Figure 4.10, except yours will not have any plan titles yet.

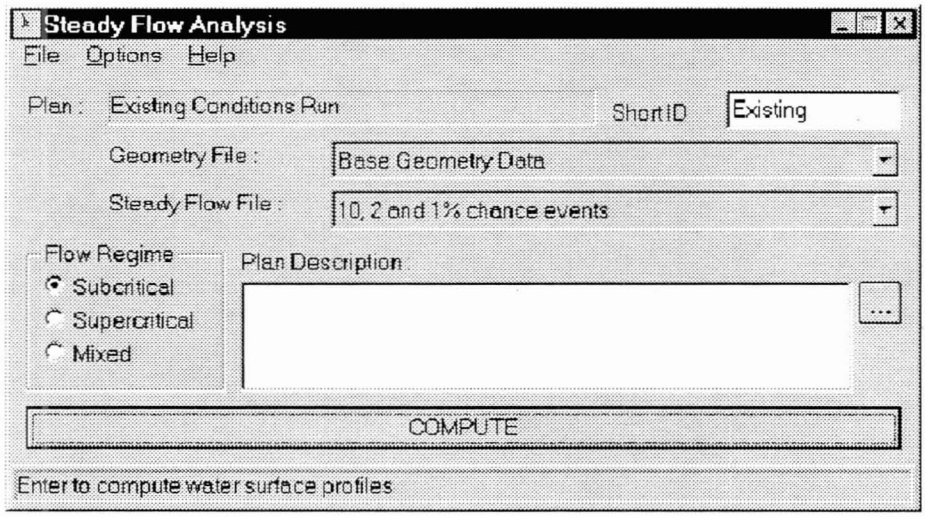


Figure 4.10 Steady Flow Analysis Simulation Window

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
- 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.

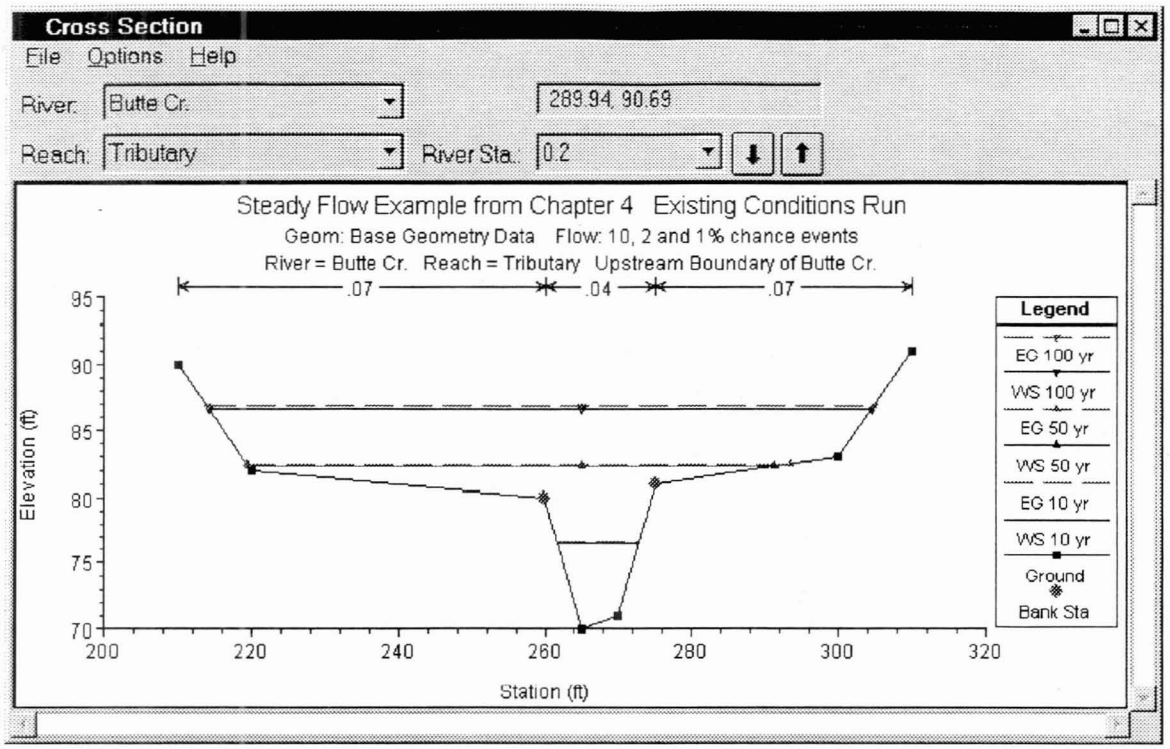


Figure 4.11 Cross Section Plot for Example Application

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.

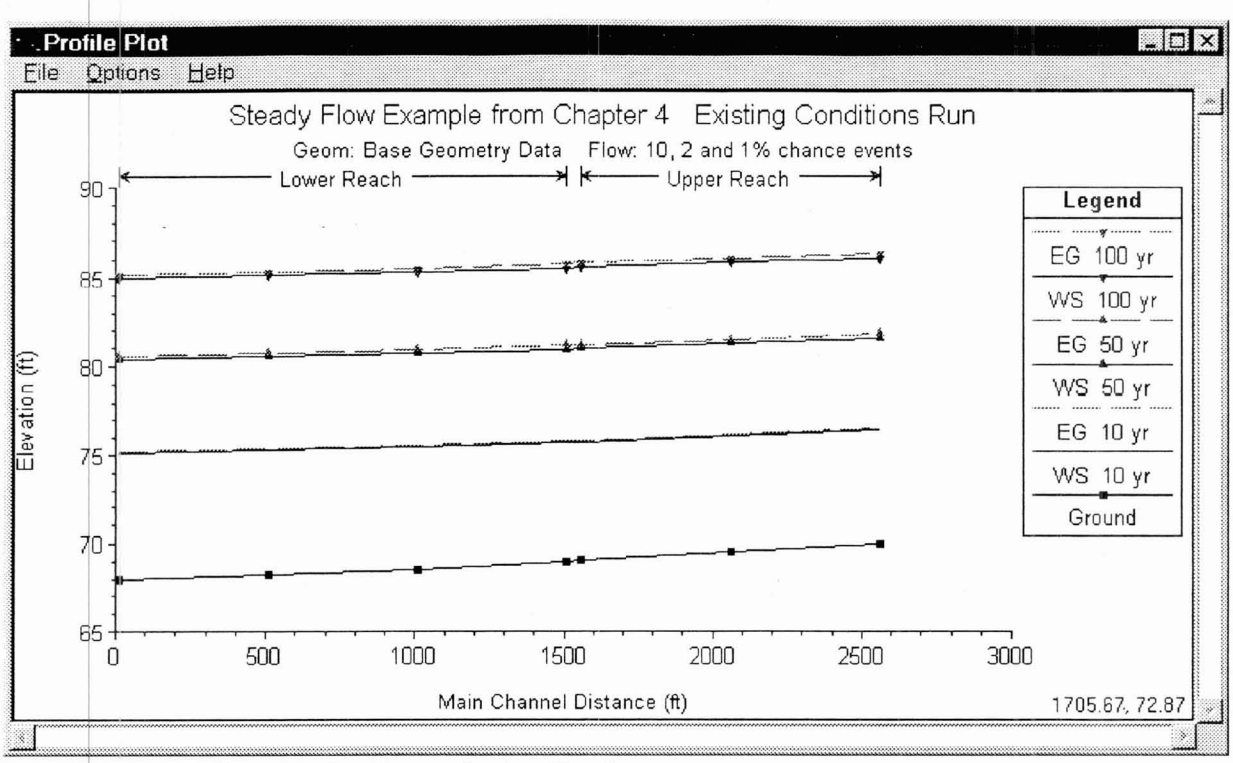


Figure 4.12 Profile Plot for Example Application

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.

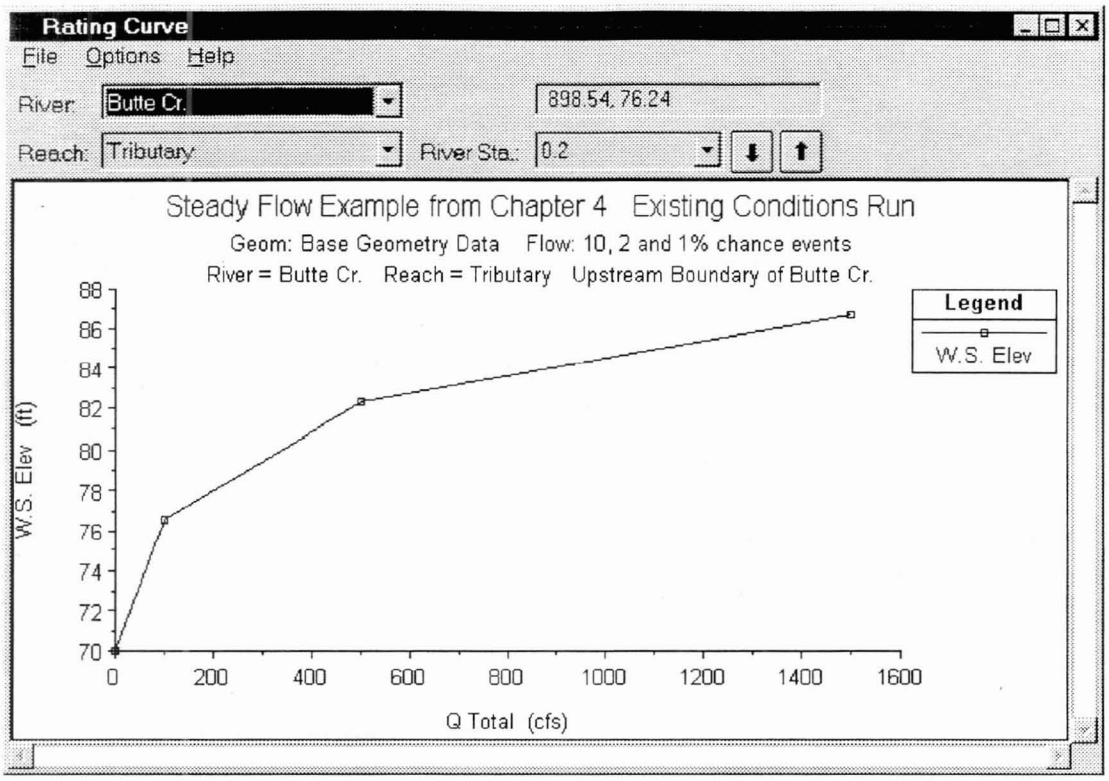


Figure 4.13 Computed Rating Curve for Example Application

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 cross section specific table and a profile table. Select **Cross Section Table** 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 table. This table shows a limited number of hydraulic variables for several cross sections. There are several types of profile tables listed under the **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.

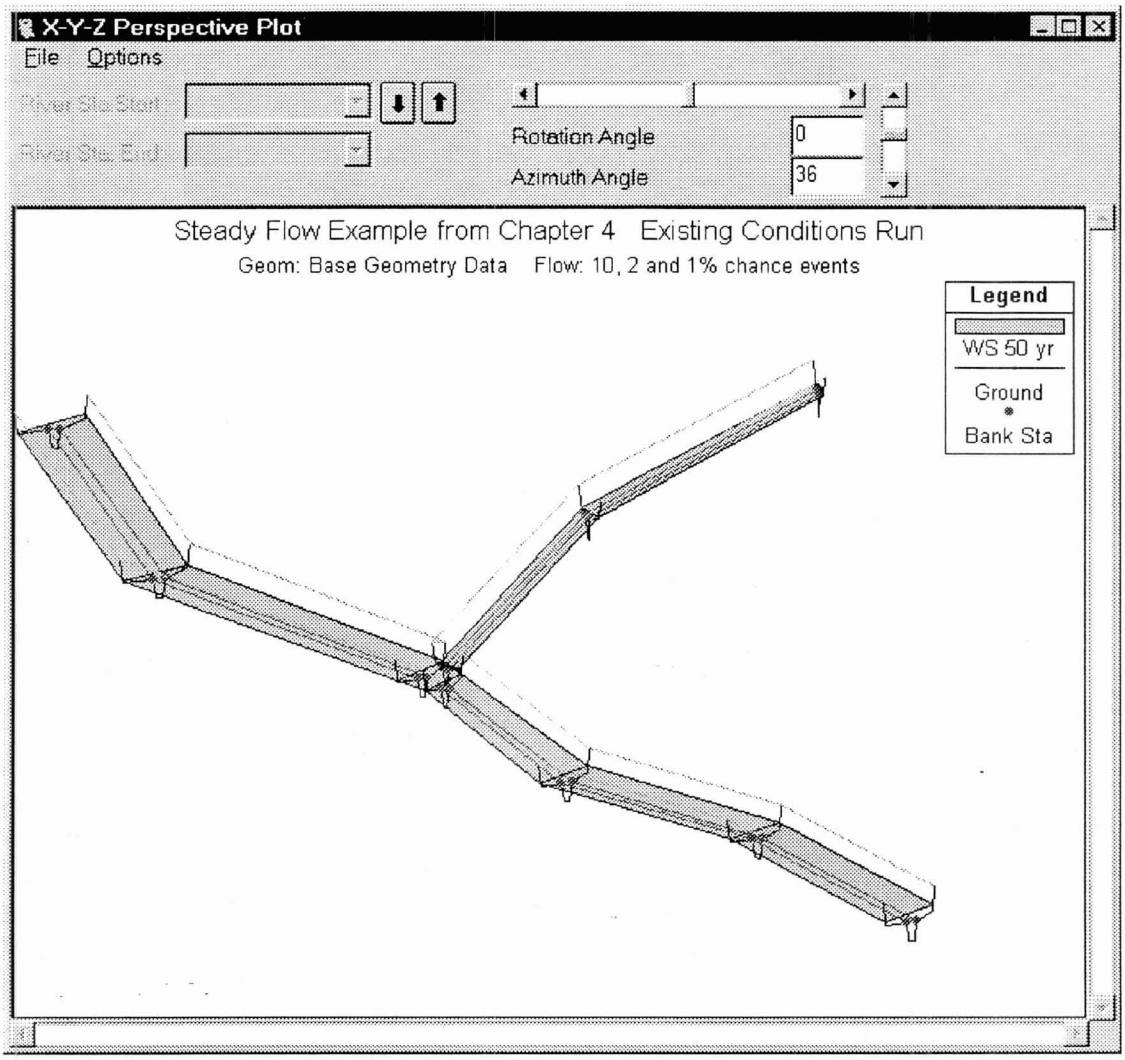


Figure 4.14 X-Y-Z Perspective Plot of All Three River Reaches

Cross Section Output

File Type Options Help

River: Fall River Profile: 10 yr

Reach: Upper Reach Riv Sta: 10

Plan: Existing River: Fall River Reach:Upper Reach Riv Sta: 10 Profile: 10 yr

W.S. Elev (ft)	76.44	Element	Left OB	Channel	Right OB
Vel Head (ft)	0.15	Wt. n-Vel.		0.035	
E.G. Elev (ft)	76.59	Reach Len. (ft)	450.00	500.00	550.00
Crit W.S. (ft)		Flow Area (sq ft)		163.31	
E.G. Slope (ft/ft)	0.000772	Area (sq ft)		163.31	
Q Total (cfs)	500.00	Flow (cfs)		500.00	
Top Width (ft)	34.86	Top Width (ft)		34.86	
Vel Total (ft/s)	3.06	Avg. Vel. (ft/s)		3.06	
Max Chl Dpth (ft)	6.44	Hydr. Depth (ft)		4.69	
Conv. Total (cfs)	17996.9	Conv. (cfs)		17996.9	
Length Wtd. (ft)	500.00	Wetted Per. (ft)		39.05	
Min Ch El (ft)	70.00	Shear (lb/sq ft)		0.20	
Alpha	1.00	Stream Power (lb/ft s)		0.62	
Frctn Loss (ft)	0.37	Cum Volume (acre-ft)		4.21	
C & E Loss (ft)	0.00	Cum SA (acres)		0.60	

Errors, Warnings and Notes

Calculated water surface from energy equation.

Figure 4.15 Detailed Tabular Output at a Cross Section

Profile Output Table - Standard Table 1

File Options Std. Tables User Tables Help

HEC-RAS Plan: Existing

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
			(cfs)	(ft)	(ft)	(ft)	(ft)	(ft/ft)	(ft/s)	(sq ft)
Fall River	Upper Reach	10	2000.00	70.00	81.61		81.84	0.000646	4.31	751.24
Fall River	Upper Reach	9.9	2000.00	69.50	81.31		81.53	0.000599	4.21	754.61
Fall River	Upper Reach	9.8	2000.00	69.10	80.97		81.22	0.000630	4.34	690.50
Fall River	Lower Reach	9.79	2500.00	69.00	80.92		81.18	0.000640	4.49	839.20
Fall River	Lower Reach	9.7	2500.00	68.50	80.76		80.92	0.000360	3.51	1094.30
Fall River	Lower Reach	9.6	2700.00	68.20	80.55		80.72	0.000400	3.73	1117.55
Fall River	Lower Reach	9.5	2700.00	68.00	80.35	74.20	80.52	0.000400	3.73	1117.53
Butte Cr.	Tributary	0.2	500.00	70.00	82.30		82.48	0.001177	3.54	189.74
Butte Cr.	Tributary	0.1	500.00	69.40	81.72		81.89	0.001164	3.52	190.91
Butte Cr.	Tributary	0.0	500.00	69.10	81.01		81.23	0.001495	3.86	162.99

Total flow in cross section

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 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.

Currently, the unsteady flow data option is not available in HEC-RAS. This option will be included in a future version.

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.

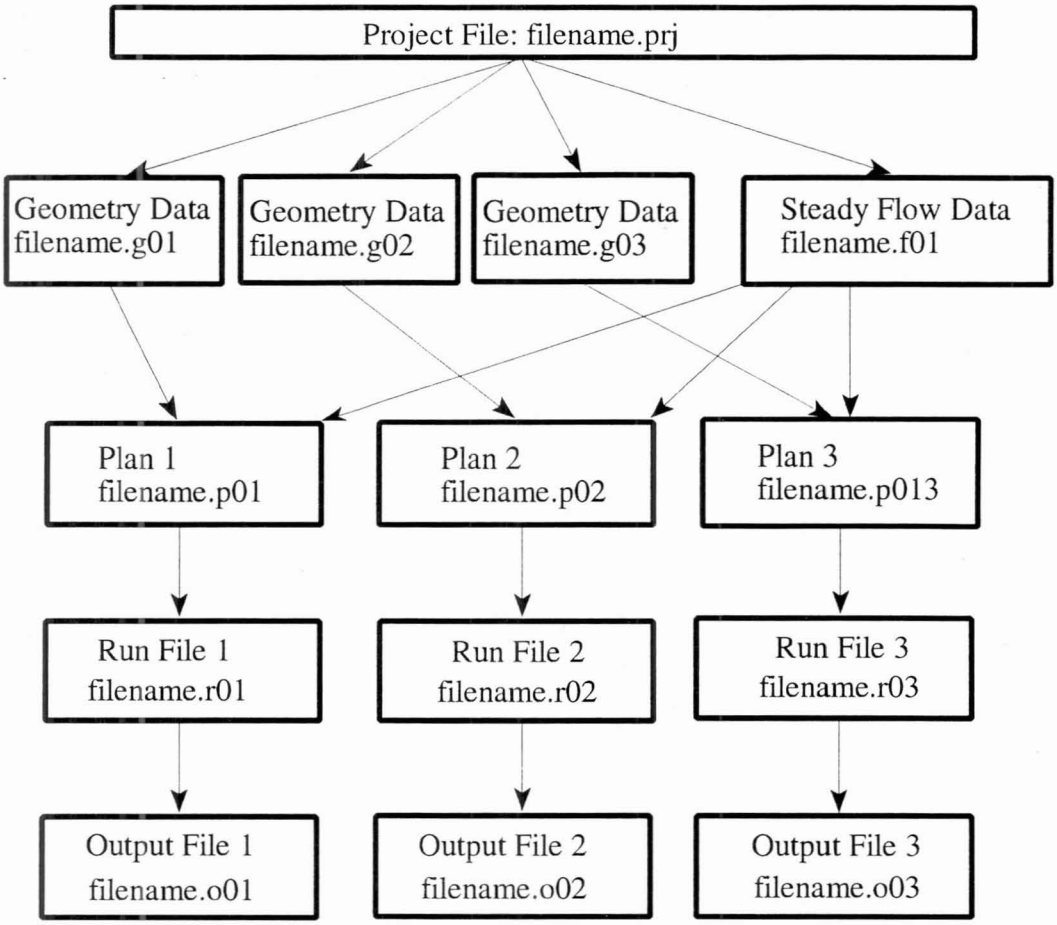


Figure 5.1 Schematic of Project Data Files.

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	
- BW to Printer	When this option is set all graphics are sent to the printer/plotter in Black and White. When this option is turned off, all graphics are sent as color drawings. Color drawings that are sent to a black and white printer will come out in grey scale shadings.
- BW to Clipboard	When this option is set all graphics are sent to the Windows Clipboard in a Black and White mode. When this option is turned off, the graphics are sent to the Clipboard as color drawings.
- Default Margins	This option allows the user to change the default margins for printing graphics and tables. The default settings are 1 inch margins on all four sides.
- 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.
- 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 Weirs and Gated Spillways
- Cross Section Interpolation
- Viewing and Editing Data Through Tables
- Importing Geometric Data
- Graphical Cross Section Editor
- Attaching and Viewing Pictures
- Saving the Geometric Data

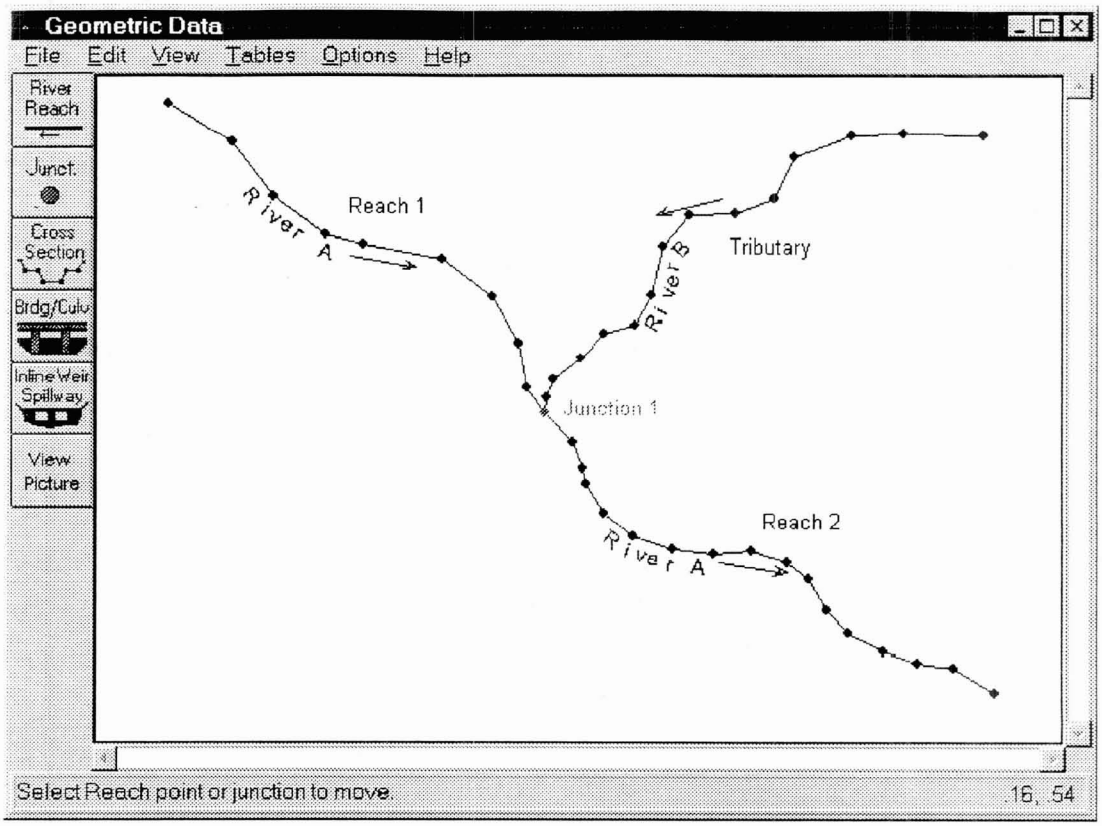


Figure 6.1 Geometric Data Window

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.

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 on the upper left side of the Geometric Data editor. 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.

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, deleting points in a reach, 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: This option allows the user to add additional points to the line that defines a reach. 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** 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" mode off, simply re-select it from the Edit menu, or select any other edit function.

Remove Points in a Reach: This option allows the user to remove points from a reach line. To use this option, first select **Remove Points in a Reach** 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.

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.

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.

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 Background Pictures: This option allows the user to turn on and off the display of any background pictures that have been loaded.

Reset Plot Extent: This option allows the user to reset the extent of the river system schematic, such that the plot will utilize the maximum amount of the available plot area.

Background Pictures

Another option available to user's is the ability to add background images for displayed behind the river system schematic. One or more pictures can be added. The user has the option of rectifying the picture by entering coordinates for the upper, lower, 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. The following graphical formats are supported for background maps: bitmap (*.bmp); icon (*.ico); windows metafile (*.wmf); GIF (*.gif); and JPEG (*.jpg).

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, and weirs).

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:

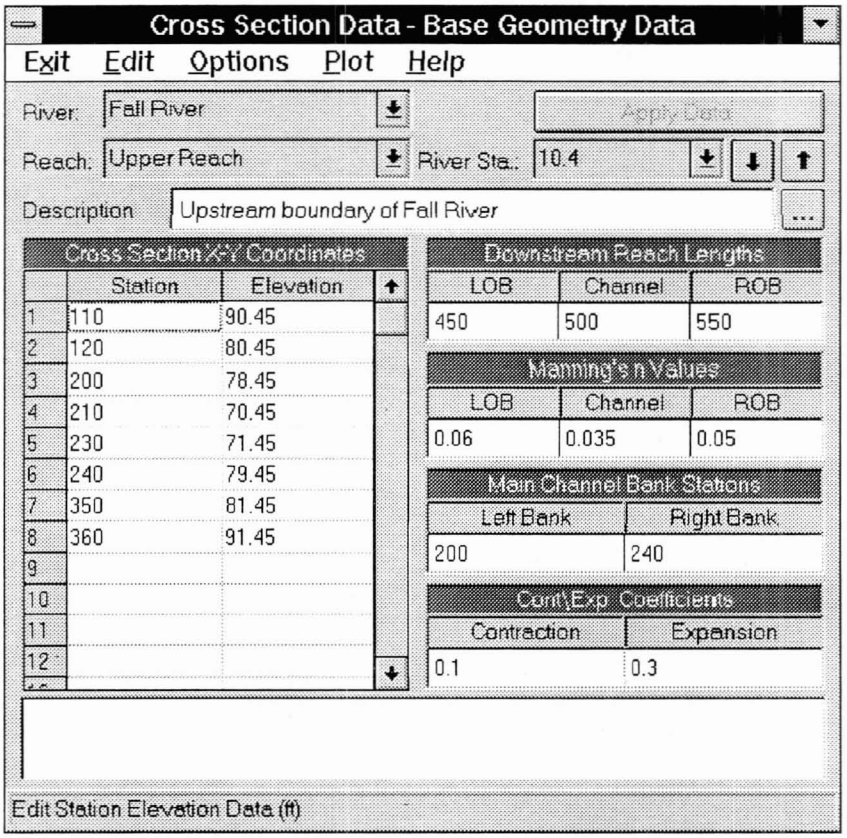


Figure 6.2 Cross Section Data Editor

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 is used as a label for cross section plots and cross section 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

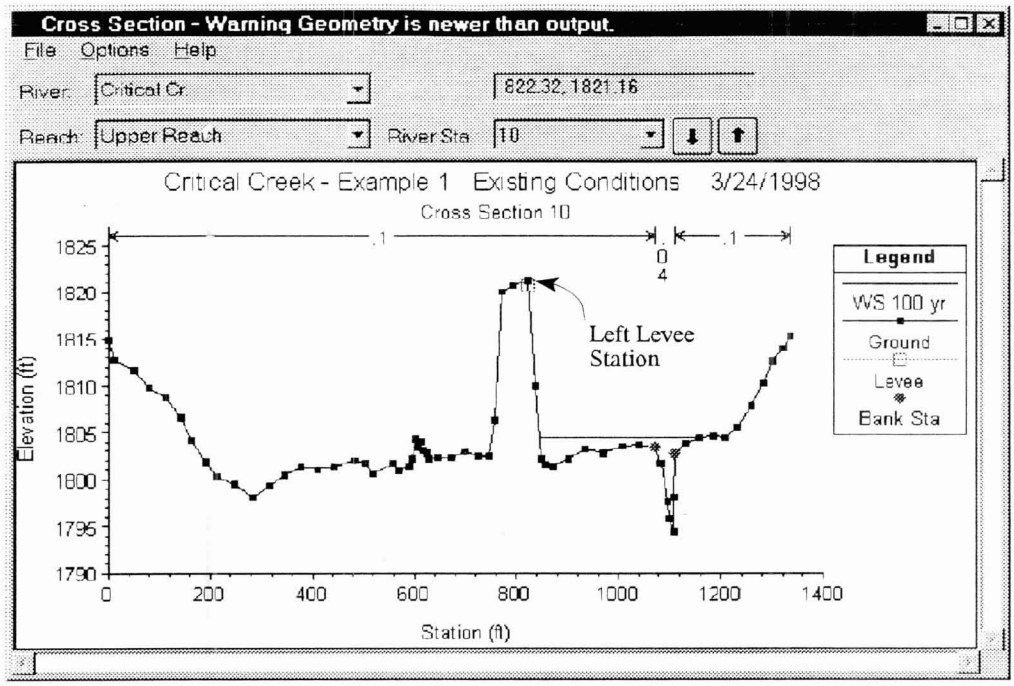


Figure 6.4 Example of the Levee Option

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.

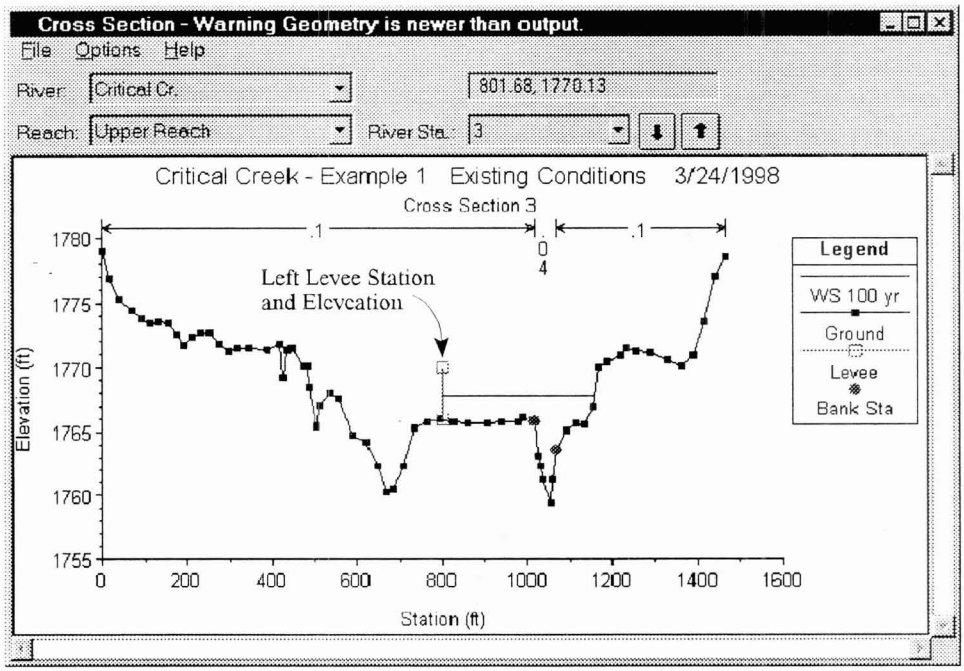


Figure 6.5 Example Levee Added to a Cross Section

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

Two alternatives are available for entering blocked obstructions. The first option allows the user to define a left station and elevation and a right station and elevation (normal blocked 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 blocked obstruction is shown in Figure 6.6.

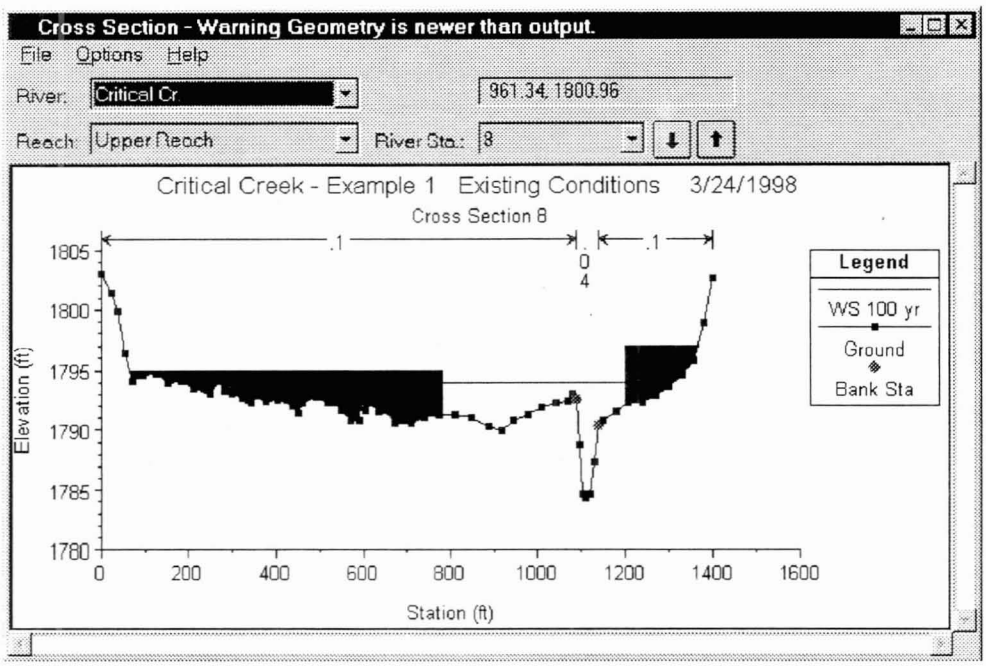


Figure 6.6 Example of Normal Blocked Obstructions

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

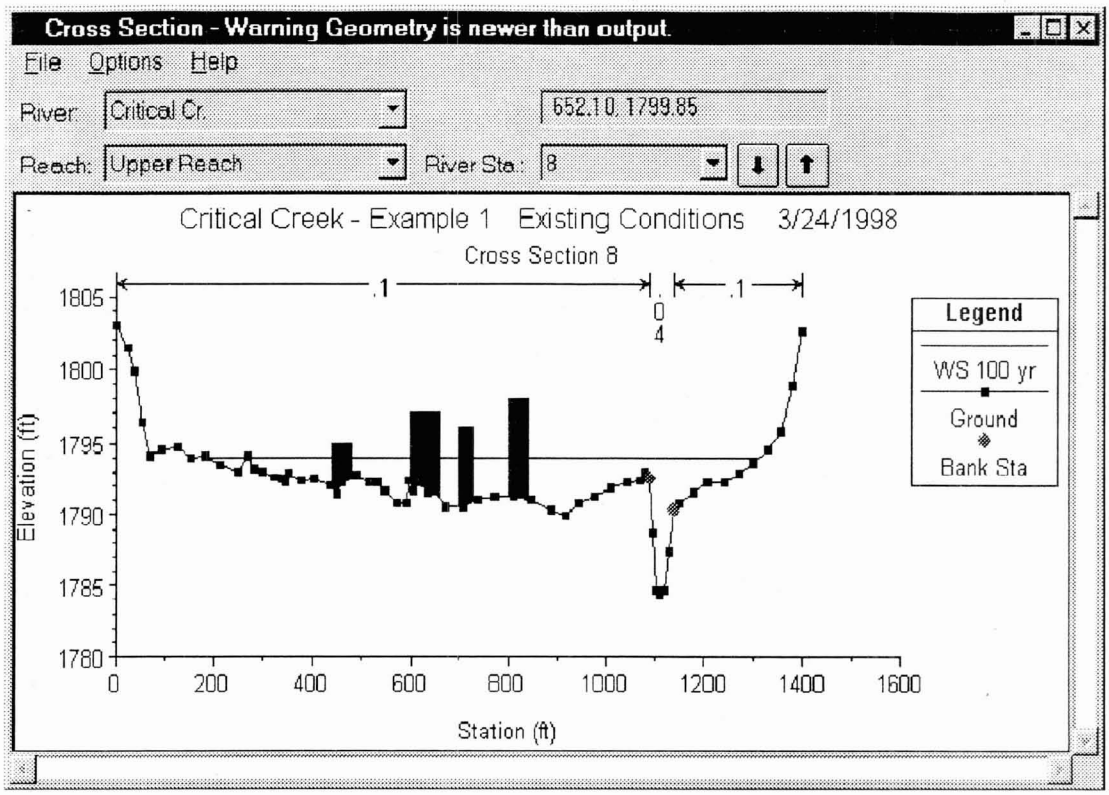


Figure 6.7 Example of a Cross Section With Blocked Obstructions

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.

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.

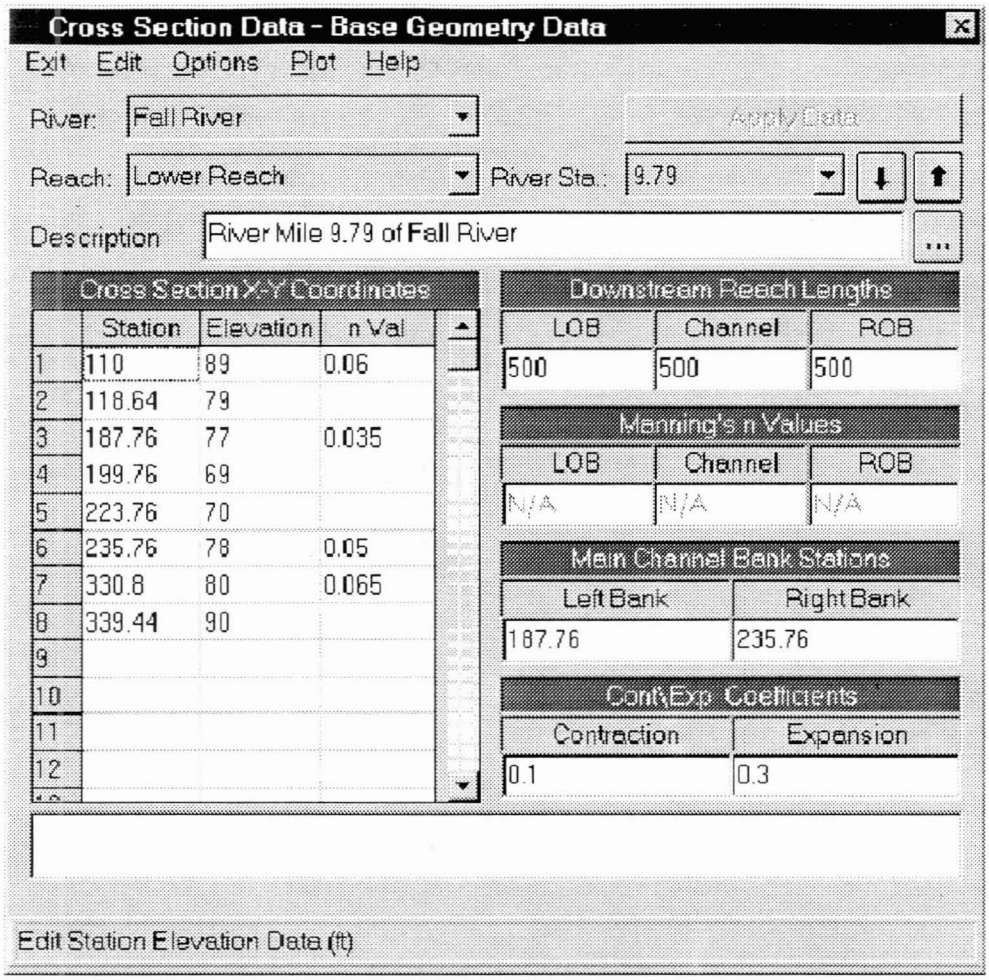


Figure 6.8 Cross section with horizontal variation of n values selected

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.

Set Maximum Sta./Elev. Points. This option allows the user to set the maximum number of station and elevation points in a cross section. The default value is set to 100 points. A maximum value of 500 can be set. This option is only necessary when the user wants to enter more than 100 points for the station and elevation points of a cross section.

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 6.9.

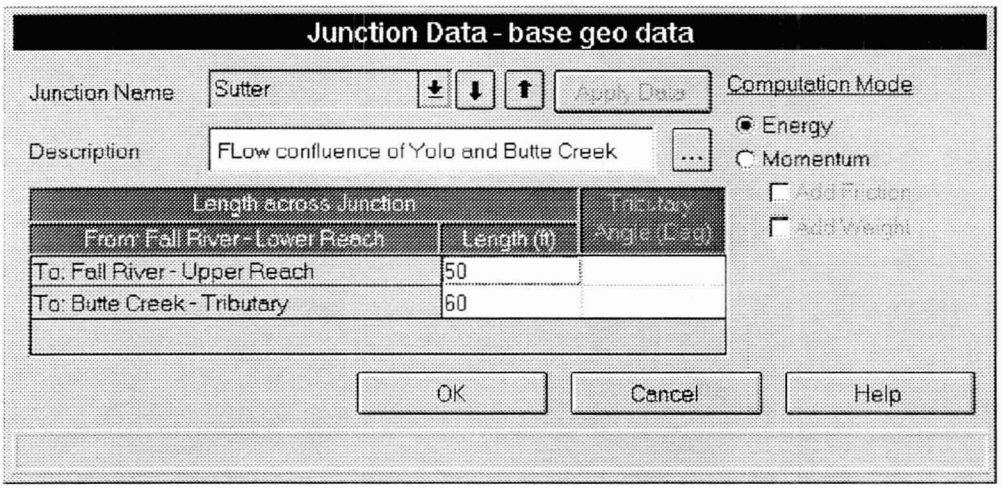


Figure 6.9 Junction Data Editor

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 6.10.

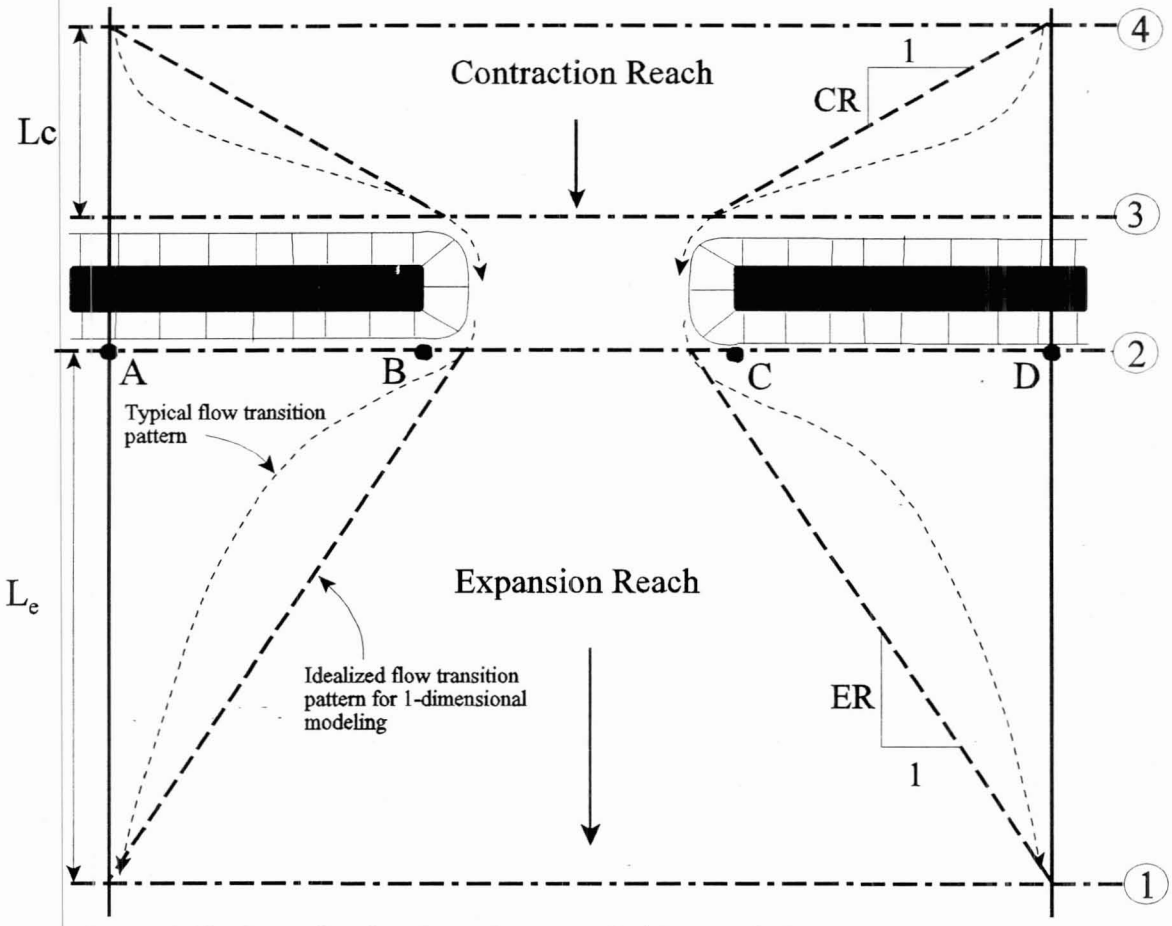


Figure 6.10 Cross Section Locations at a Bridge or Culvert

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. If field investigation is not possible, then there are two sets of criteria for locating the downstream section. The USGS has established a criterion of locating cross section 1 a distance downstream from the bridge as equal to one times the bridge opening width. Traditionally the Corps of Engineers criterion has been to locate the downstream cross section about four times the average length of the side constriction caused by the structure abutments. In practical applications this 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. Recently a detailed study was completed by the Hydrologic Engineering Center entitled "Flow Transitions in Bridge Backwater Analysis" (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.

Cross section 2 is located immediately downstream from the bridge (i.e., within a few feet). This cross section should represent the effective flow just outside the bridge.

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 effective flow area just upstream of the bridge.

Both cross sections 2 and 3 will have ineffective flow areas to either side of the bridge opening during low flow and pressure flow profiles. 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 HEC-RAS 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. The USGS has established a criterion for locating cross section 4 a distance upstream from the bridge equal to one times the bridge opening width (the distance between points B and C on Figure 6.10). Traditionally, the Corps of Engineers used a criterion to locate the upstream cross section 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.

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.

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 considered 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 available for computing losses through the bridge (sections 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 equation 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 6.11. To add a bridge to the model the user must do the following:

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

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

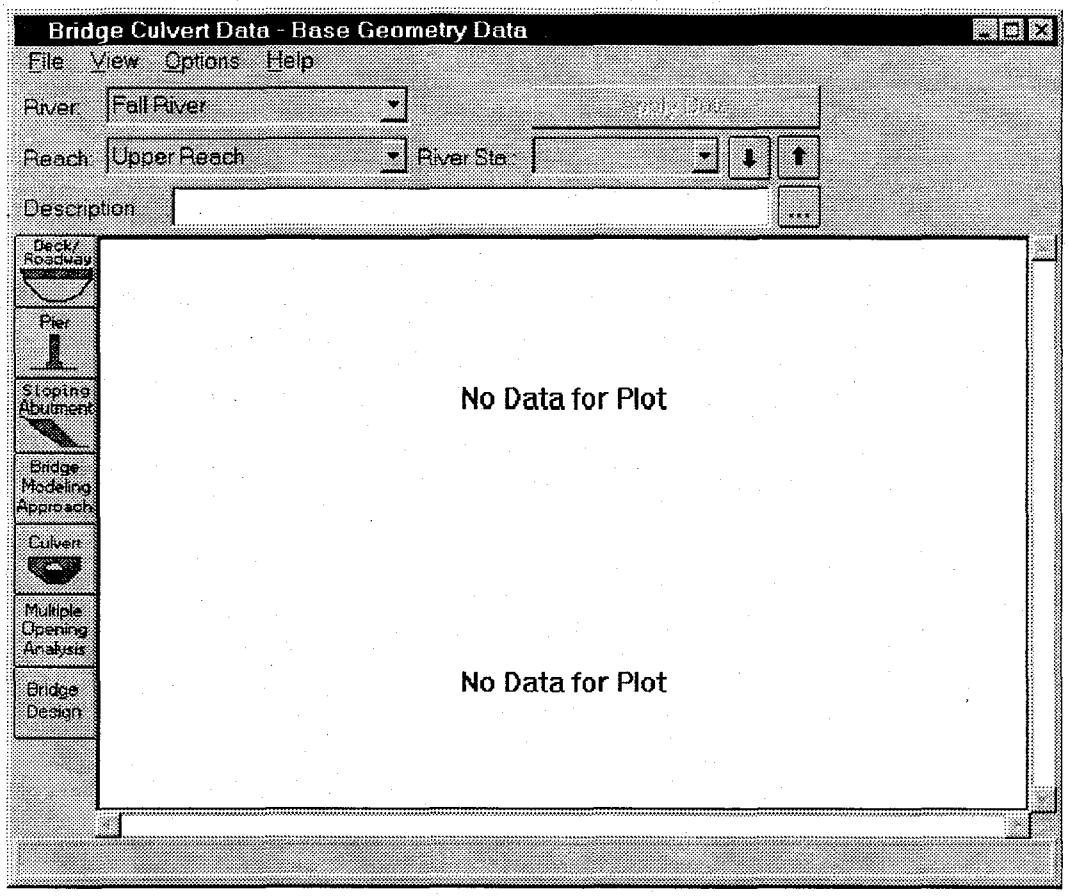


Figure 6.11 Bridge/Culvert Data Editor.

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 6.12 (except yours will be blank). The information entered in the deck editor consists of the following:

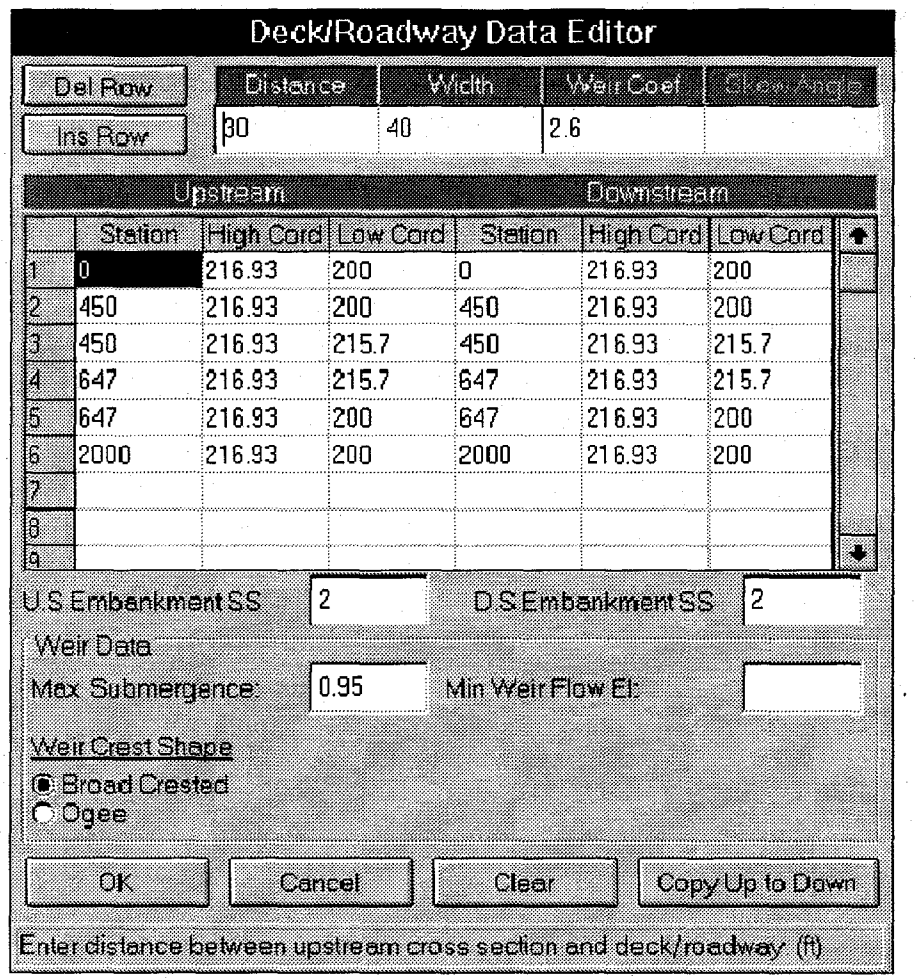


Figure 6.12 Bridge Deck/Roadway Data Editor

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. 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. 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).

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

Skew Angle - Angle that the bridge deck is skewed from a line perpendicular to the flow lines passing through the bridge.

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.

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 6.13. **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.

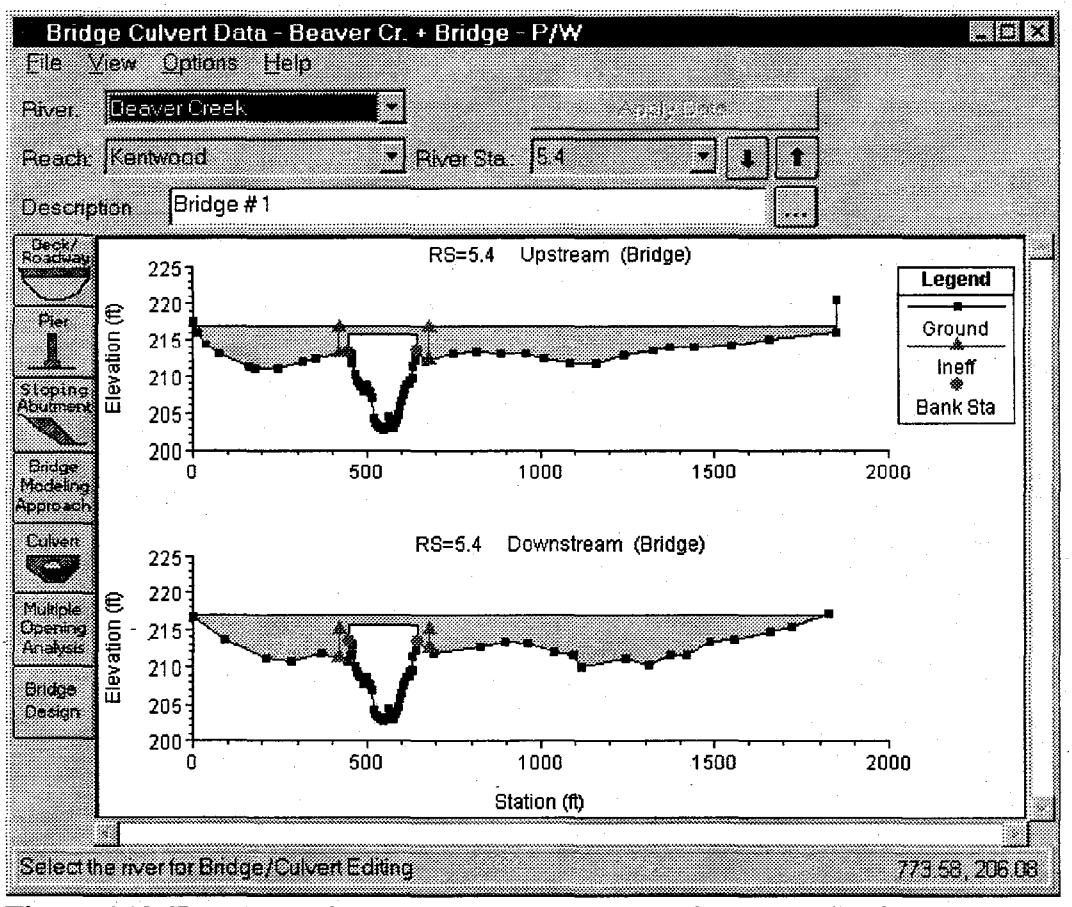


Figure 6.13 Example Bridge Deck Plotted on Bounding Cross Sections

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 6.14.

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.

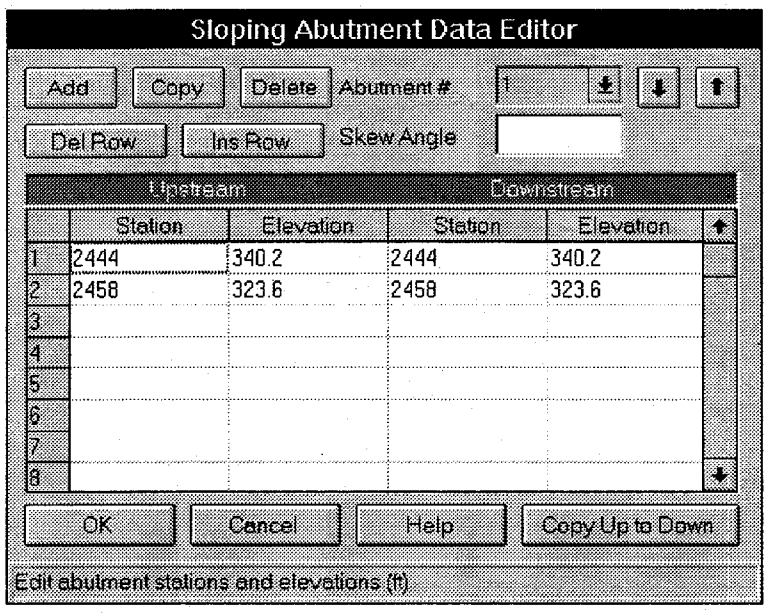


Figure 6.14 Abutment Data editor

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 6.15. This graphic is zoomed in on the left abutment of the bridge.

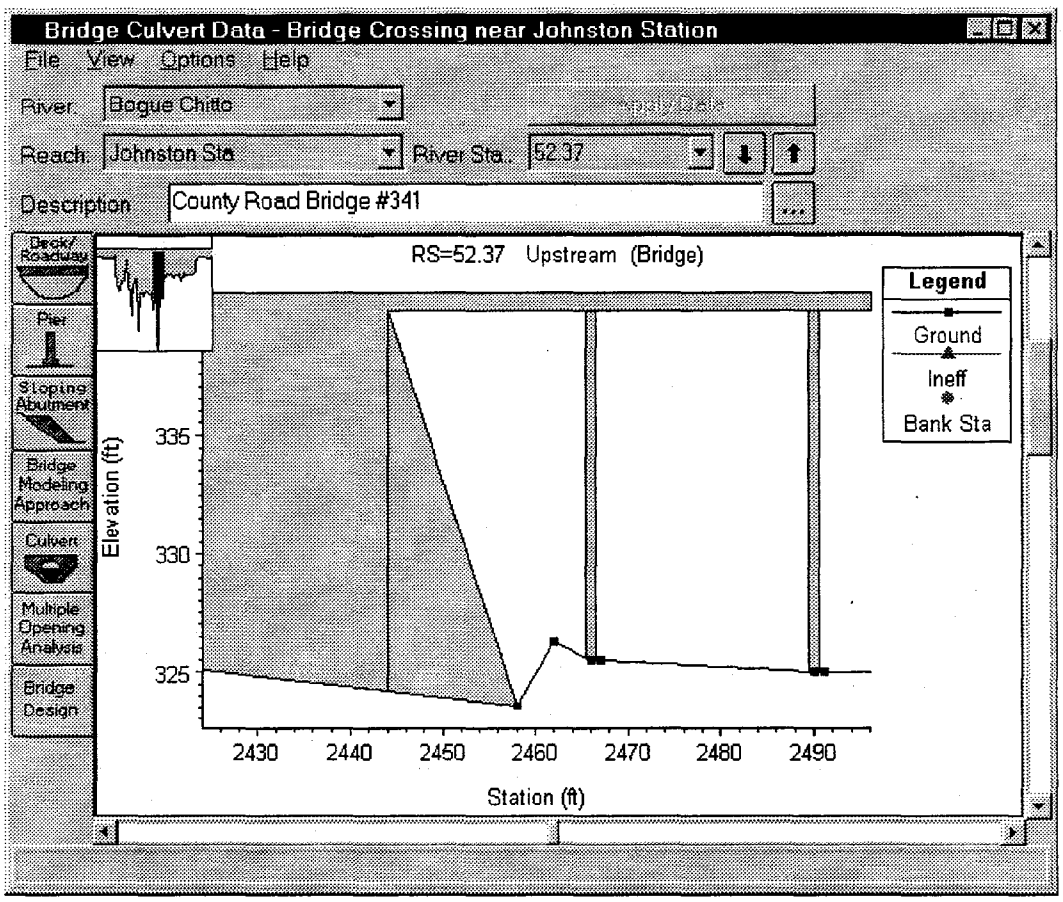


Figure 6.15 Example of a Sloping Abutment

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.

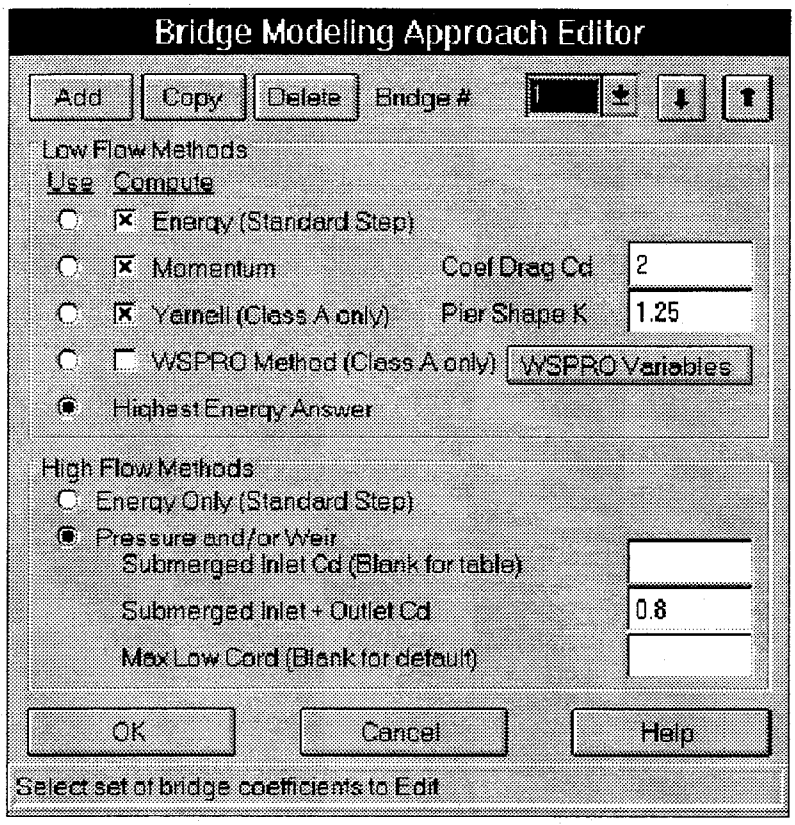


Figure 6.18 Bridge Modeling Approach Editor

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 6.19 will appear.

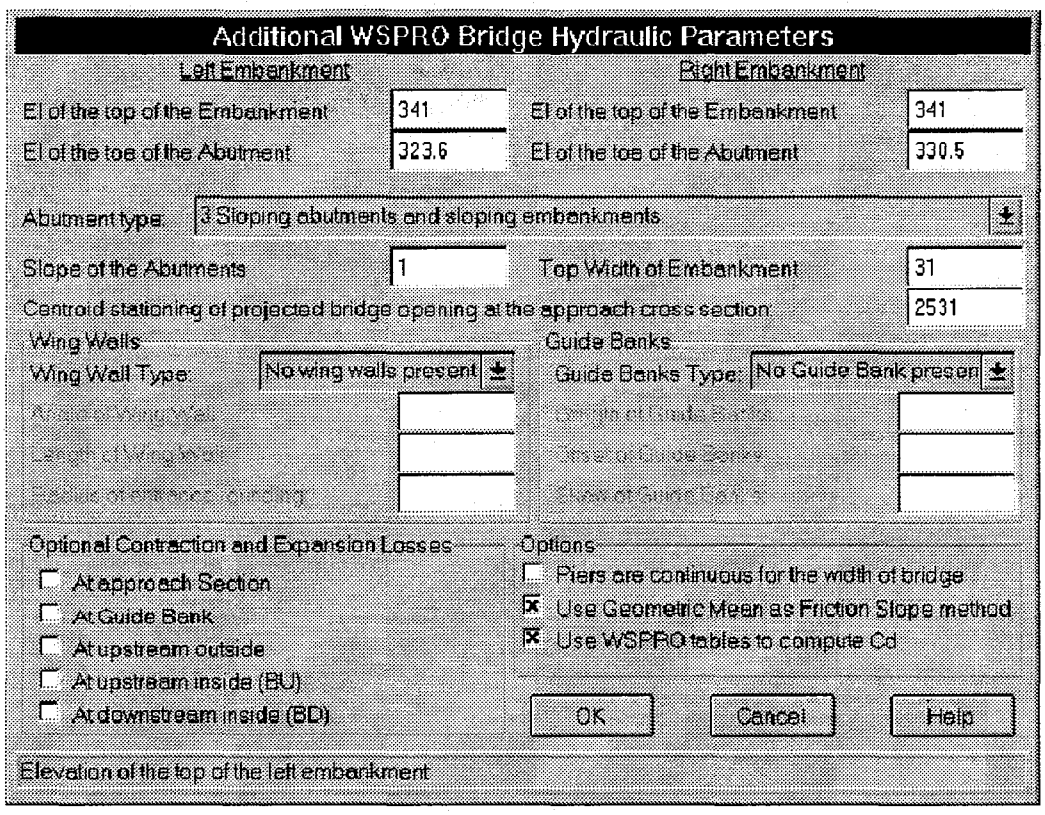


Figure 6.19 WSPRO Data Editor

As shown in Figure 6.19, 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. User's 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 submerged. Generally this coefficient is around 0.8. For more information on pressure flow coefficients see Hydraulics of Bridge Waterways (FHWA, 1978).

Max Low Chord - 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 6.20. 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).

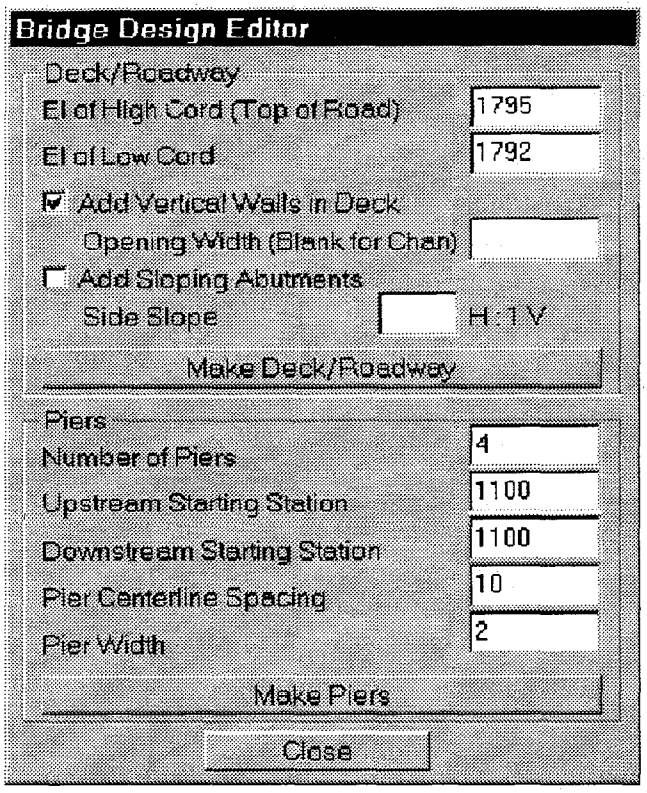


Figure 6.20 Bridge Design Editor

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 eight different types of culvert shapes. These shapes include box (rectangular), circular, elliptical, arch, pipe arch, semi circular, low-profile arch, and high-profile arch 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 6.11). To add a culvert group to the model the user must then do the following:

1. Select the river and reach that you would like to place the culvert in. This is accomplished by pressing the down arrow on the 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 6.21 (except yours will be blank). The information entered in the Culvert Data Editor consists of the following:

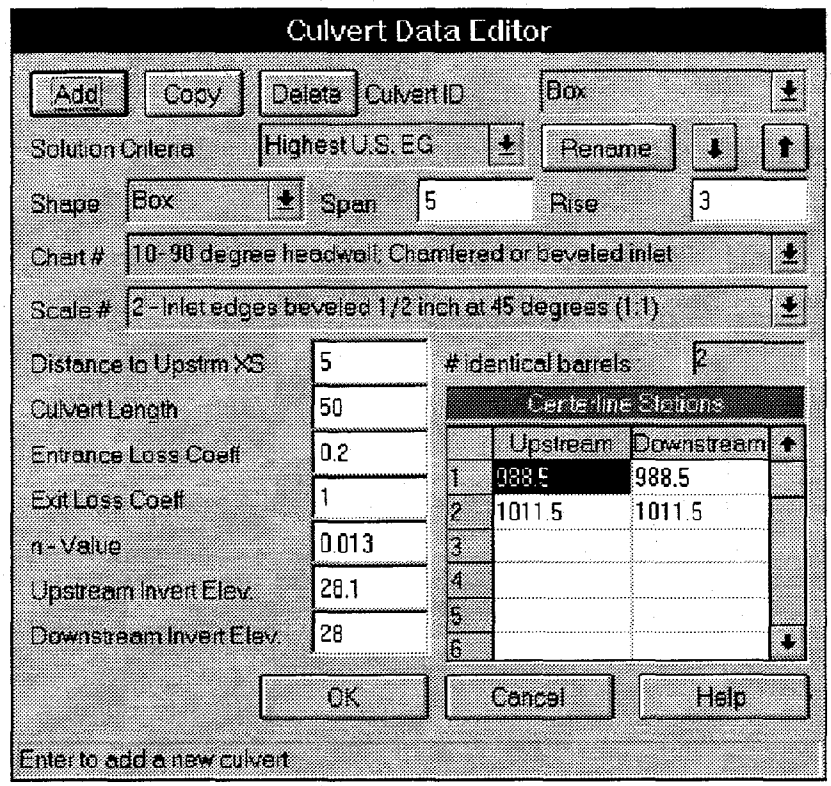


Figure 6.21 Culvert Data Editor

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 eight available shapes. This is accomplished by pressing the down arrow on the side of the box, and then selecting one of the eight available shapes.

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 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.

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.

n-value - The n-value field is used for entering the Manning's n value of the culvert barrel.

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.

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 6.22. **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.

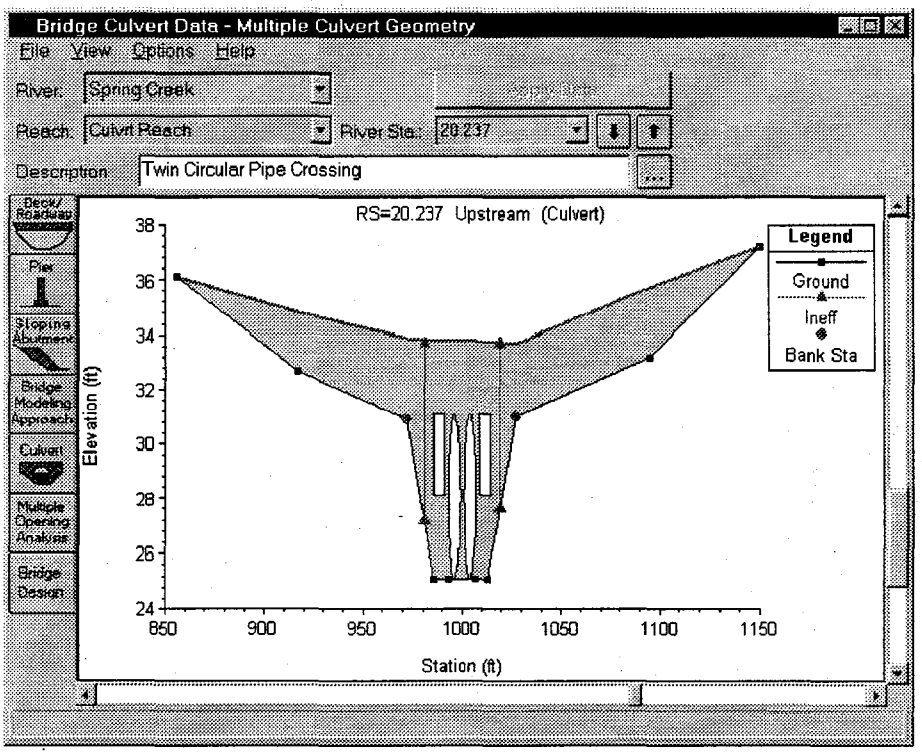


Figure 6.22 Bridge/Culvert Data Editor with example culvert

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.

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 re-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.

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 are piers 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 6.23. 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.

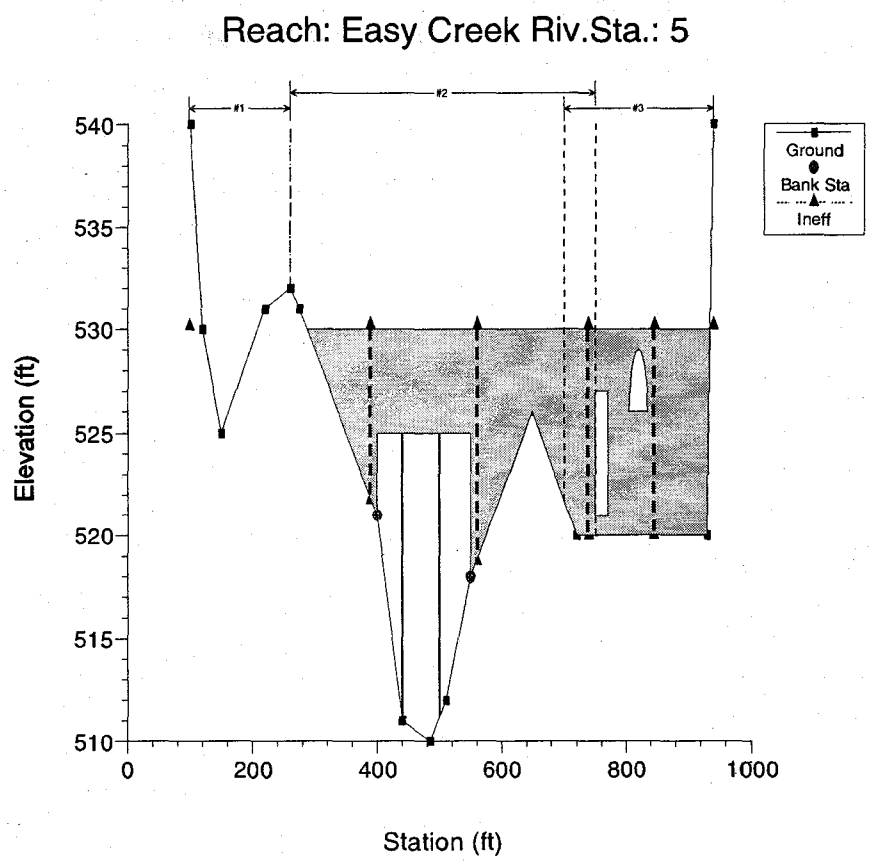


Figure 6.23 Example Multiple Opening River Crossing

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 grey shaded area in Figure 6.23, 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.

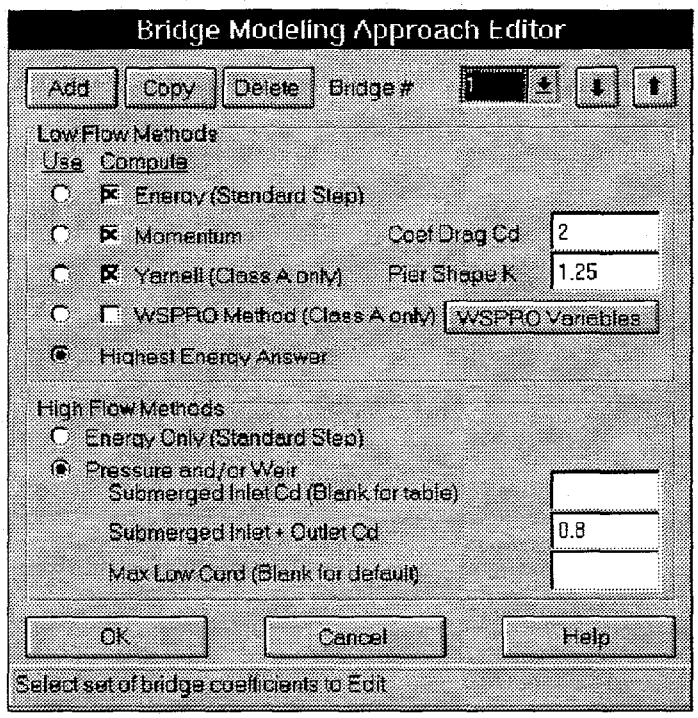


Figure 6.24 Bridge Modeling Approach Editor

As shown in Figure 6.24, 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 6.25 (except yours will be blank the first time you bring it up).

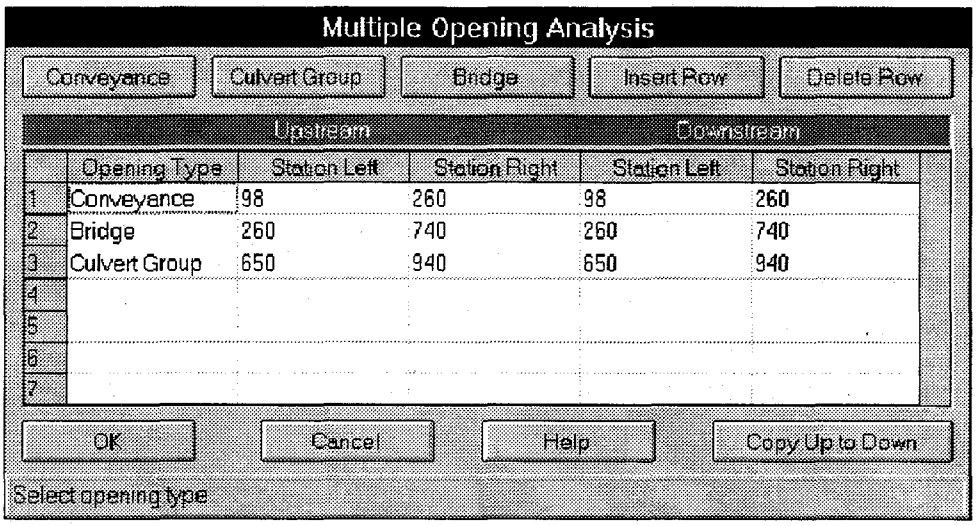


Figure 6.25 Multiple Opening Analysis window

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 6.25 (numerical representation) and Figure 6.23 (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 through this area will be solved 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, theprogram 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 analysis can be found in Chapter 7 of the HEC-RAS Hydraulic Reference manual.

Inline Weirs and Gated Spillways

The current version of HEC-RAS allows the user to model inline (across the main stream) weirs and gated spillways. Lateral weirs and gated spillways are not available in this version, but will be added to a future version of the software. HEC-RAS has the ability to model radial gates (often called tainter gates) or vertical lift gates (sluice gates). The spillway crest of the gates can be modeled as either an ogee shape or a broad crested weir shape. In addition to the gate openings, the user can also define a separate uncontrolled overflow weir.

This section of the User's manual will describe how to enter the data for inline weirs and gated spillways. 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 weir and gated spillway, see Chapter 8 of this User's manual.

Entering and Editing Inline Weir and Gated Spillway Data

Inline weir and gated spillway data are entered in a similar manner as bridge and culvert data. To enter an inline weir and/or gated spillway press the **Inline Weir/Spillway** button from the Geometric Data window. Once this button is pressed, the Inline Weir and Gated Spillway Data editor will appear as shown in Figure 6.26 (except yours will be blank until you have entered some data).

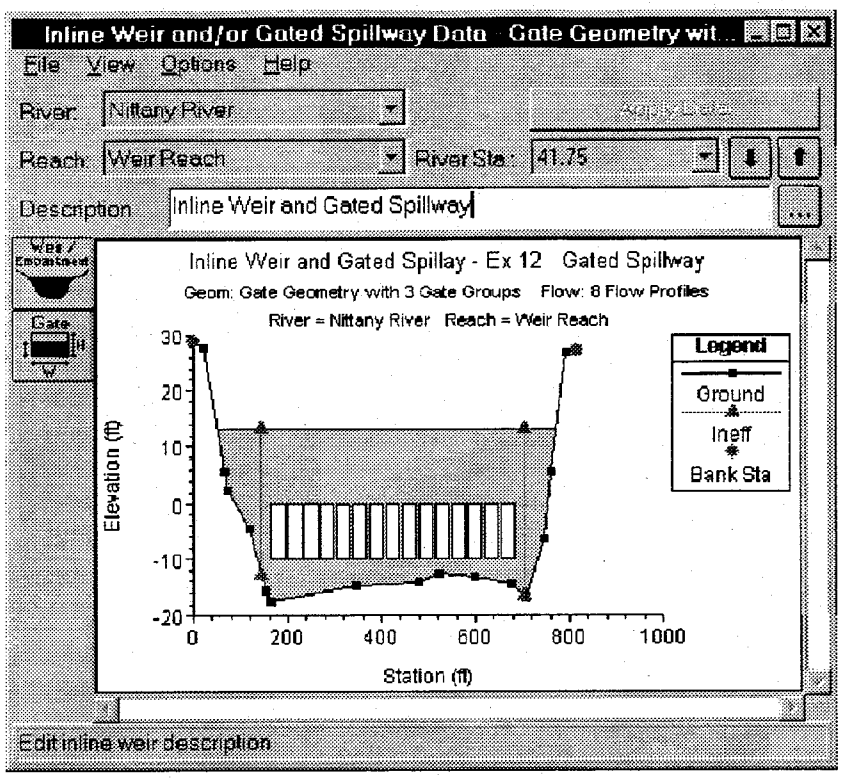


Figure 6.26 Inline Weir and Gated Spillway Data Editor

To add an inline weir and/or gated spillway to a model, the user must do the following:

1. Select the river and reach that you would like to place this inline weir and/or spillway 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 Weir and/or Gated Spillway Data editor.

2. Go to the **Options** menu at the top of the window and select **Add an Inline Weir and/or Gated Spillway** 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 weir and/or gated spillway crossing.

3. Enter all of the data for the Inline Weir and/or Gated Spillway. 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 Weir and/or Gated Spillway 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 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 inline weir and/or gated spillway should be numerically between the two cross sections that bound the structure. Once the user selects **Add an Inline Weir and/or Gated Spillway** 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 Weir and/or Gated Spillway will be displayed on the editor.

Description. The description box is used to describe the Inline Weir and/or Gated Spillway 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 Weir and/or Gated Spillway plots and tables.

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 as shown in Figure 6-27.

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).

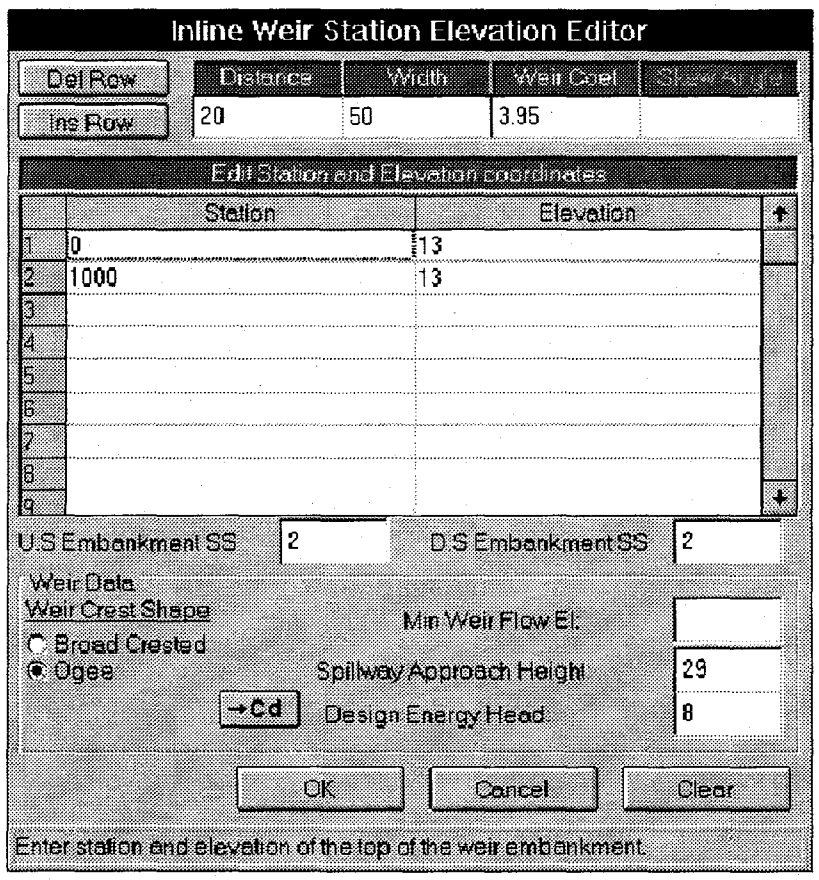


Figure 6.27 Weir and Embankment Data Editor

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 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.

Skew Angle - Angle that the weir/embankment is skewed from a line perpendicular to the flow lines.

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.

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 weir/embankment, and are not effected by this elevation. If this field is left blank, the elevation that triggers weir flow is based on the lowest elevation of the station and elevation coordinates.

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 require. 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 **Cd** 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 Weir and Gated Spillway data editor. Once this button is pressed, the gated editor will appear as shown in Figure 6.28 (except yours will be blank until you have entered some data).

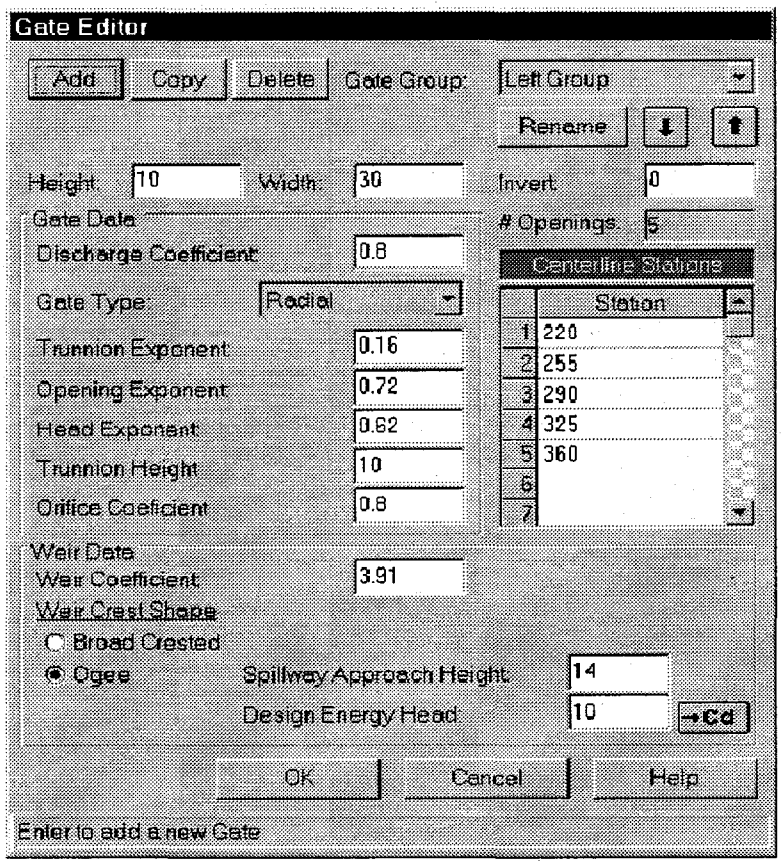


Figure 6.28 Gated Spillway Editor

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 for metric).

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. A typical value for a radial gate is 0.16. If you have selected a gate type of sluice gate, this value will be set to 0.0 and should not be changed.

Opening Exponent - This field is used to enter the gate opening exponent, which is used in the radial gate equation. A typical value for a radial gate is 0.72. If you have selected a gate type of sluice gate, this value will be set to 1.0 and should not be changed.

Head Exponent - This field is used to enter the upstream energy head exponent, which is used in the radial gate equation. A typical value for a radial gate is 0.62. If you have selected a gate type of sluice gate, this value will be set to 0.5, which is a normal value for a sluice gate.

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.

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 the Spillway Approach Height and the Design Energy Head, both are explained below. Once these fields are entered, the user should press the button labeled **Cd**. 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.

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 **Options** menu bar as shown in Figure 6.29.

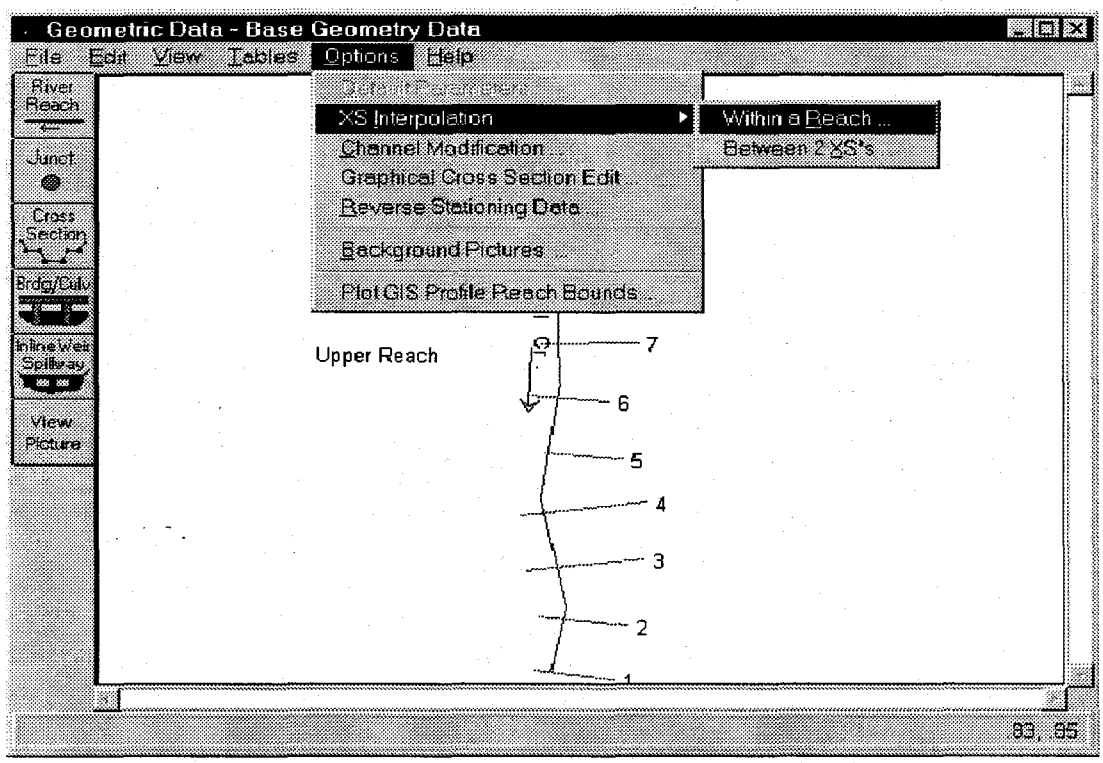


Figure 6.29 Automatic Cross Section Interpolation Options

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 6.30. 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.

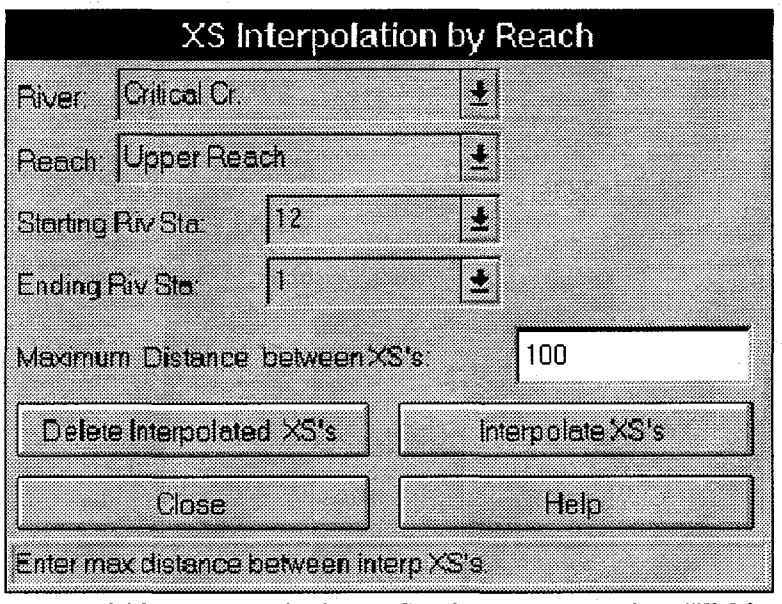


Figure 6.30 Automatic Cross Section Interpolating Within a Reach

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 6.31.

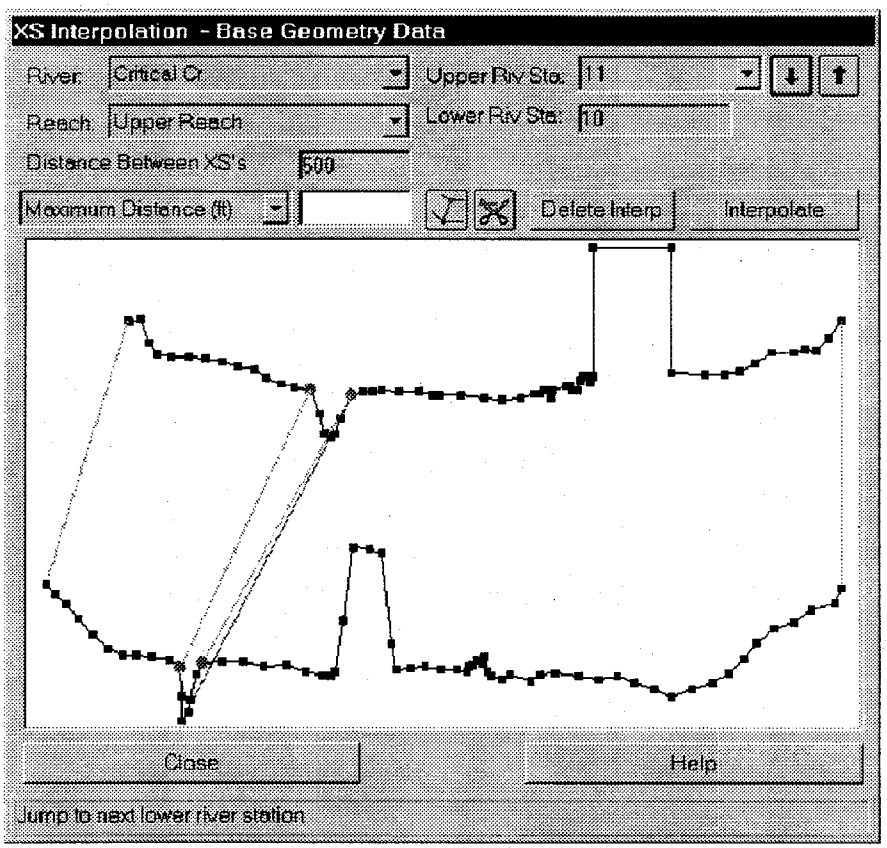


Figure 6.31 Detailed Cross Section Interpolation Window

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 6.31. 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 6.31, 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 in Figure 6.32.

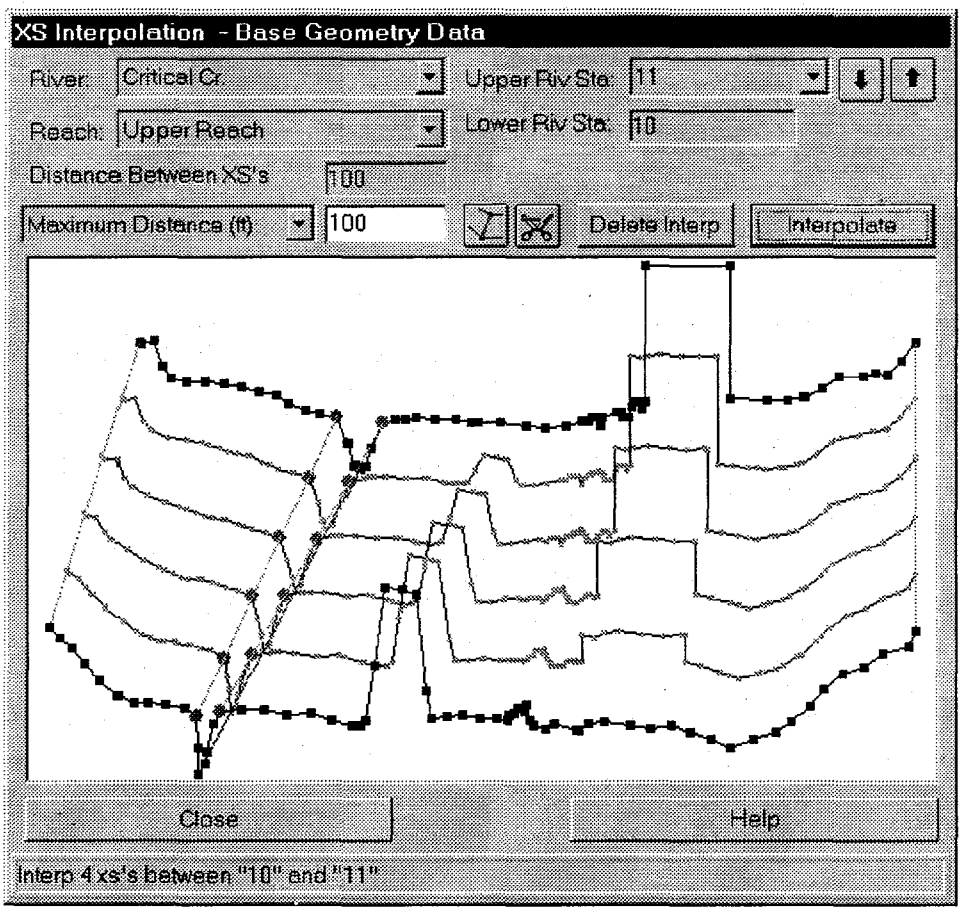


Figure 6.32 Cross Section Interpolation Based on Default Master Cords

As shown in Figure 6.32, 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 top 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 6.33.

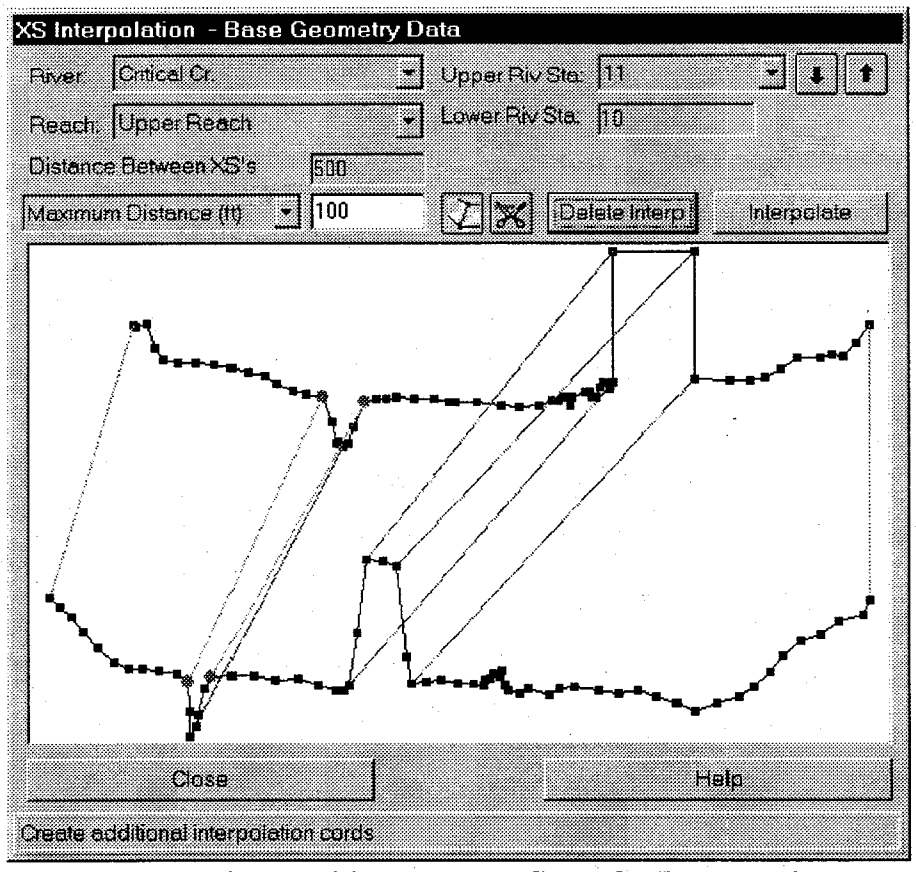


Figure 6.33 Adding Additional Master Cords for Interpolation

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 6.34.

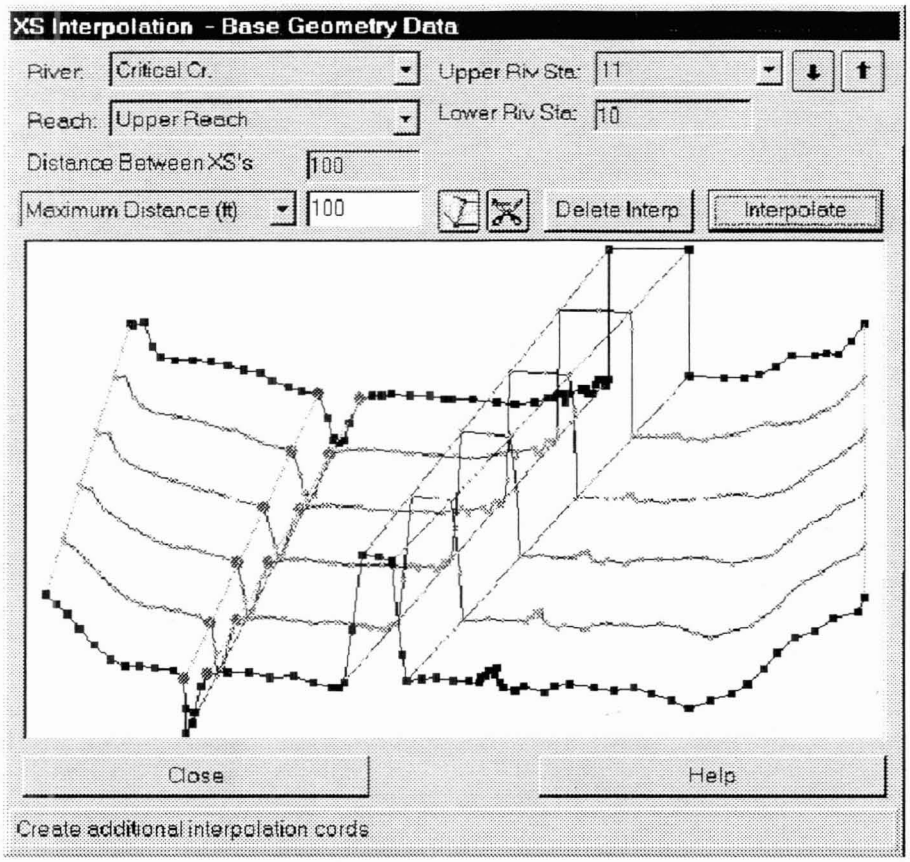


Figure 6.34 Final Interpolation With Additional Master Cords

As shown in Figure 6.34, 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 sections can then be developed by adding additional master cords in order to 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 8 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 6.35). All ice data for this cross section can be entered with this editor.

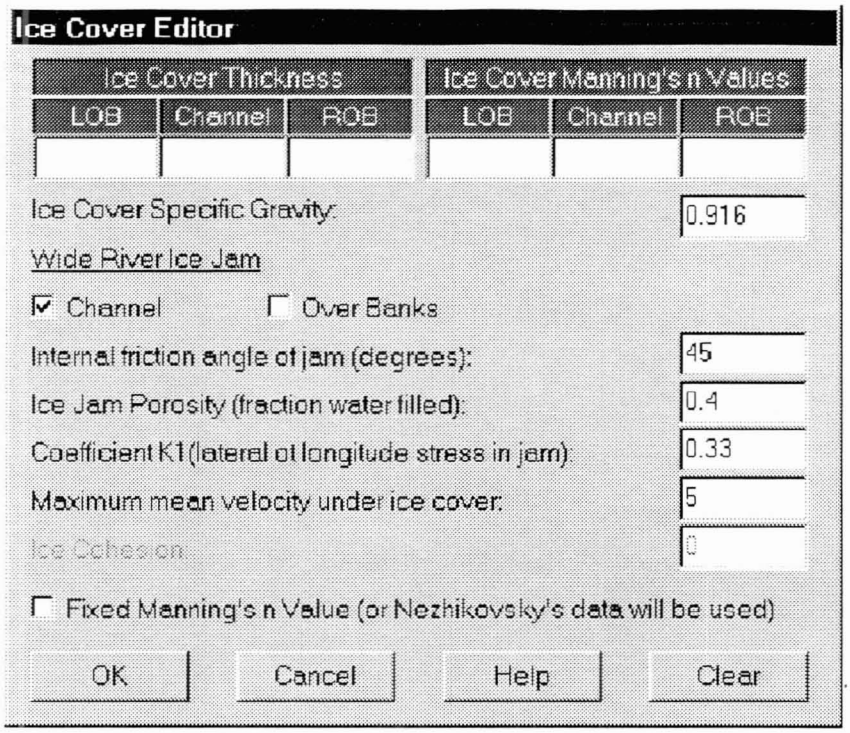


Figure 6.35 Ice Cover Editor

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.

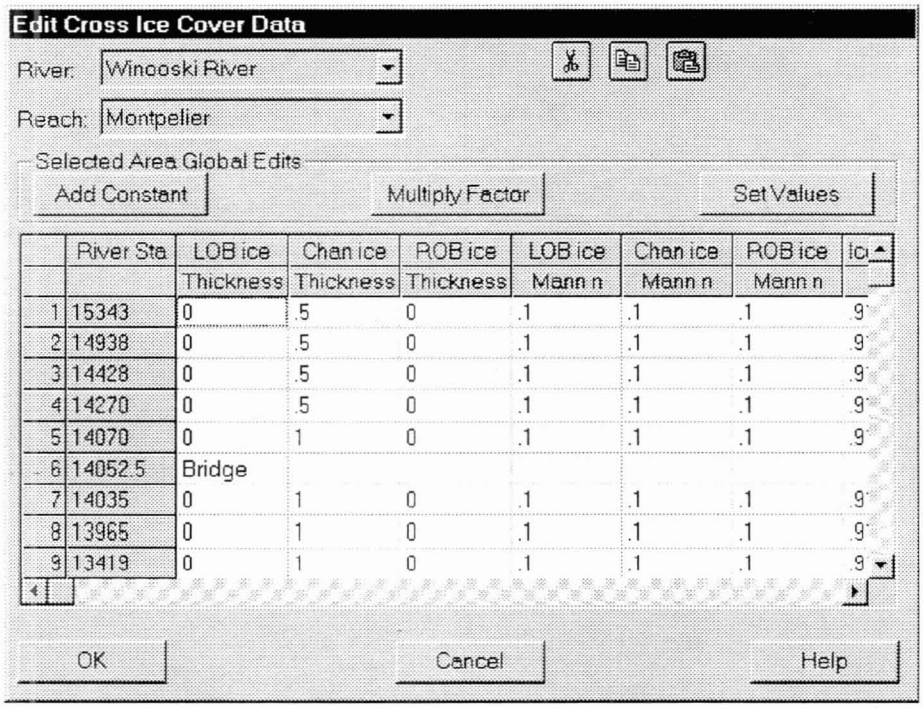
Ice Cohesion. At present, the ice jam cohesion is set to the default value of zero. This cannot be changed by the user. A value of zero is appropriate for breakup ice jams.

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 6.36).



	River Sta	LOB ice Thickness	Chan ice Thickness	ROB ice Thickness	LOB ice Mann n	Chan ice Mann n	ROB ice Mann n	ic
1	15343	0	.5	0	.1	.1	.1	.9
2	14938	0	.5	0	.1	.1	.1	.9
3	14428	0	.5	0	.1	.1	.1	.9
4	14270	0	.5	0	.1	.1	.1	.9
5	14070	0	1	0	.1	.1	.1	.9
6	14052.5	Bridge						
7	14035	0	1	0	.1	.1	.1	.9
8	13965	0	1	0	.1	.1	.1	.9
9	13419	0	1	0	.1	.1	.1	.9

Figure 6.36 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 6.37.

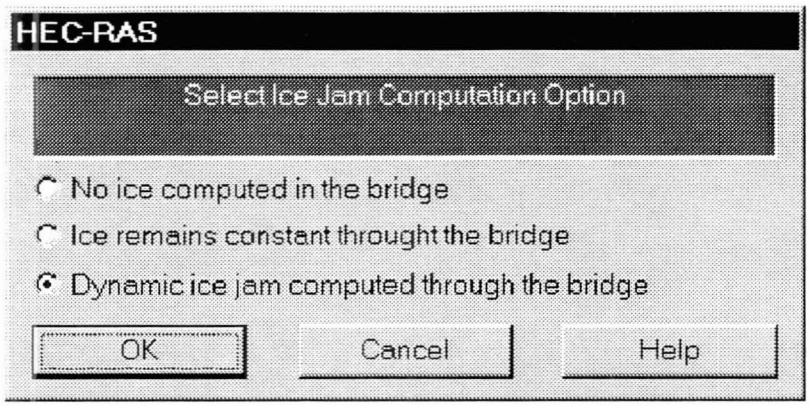


Figure 6.37 Entering ice information at bridges

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

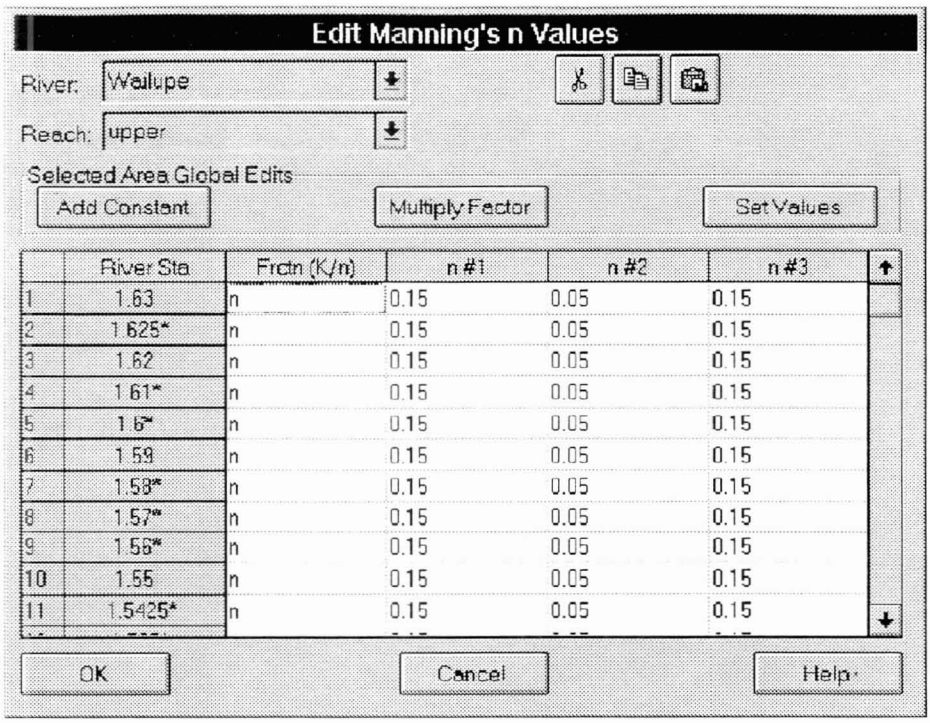
Once cross section data are entered, the user can view and edit certain types of data in a tabular format. The current version of HEC-RAS allows the user to view and edit Manning's n or k values, cross section reach lengths, contract and expansion coefficients, and the cross section river stationing. 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 6.38.

As shown in Figure 6.38, 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



	River Sta	Frctn (K/n)	n #1	n #2	n #3
1	1.63	n	0.15	0.05	0.15
2	1.625*	n	0.15	0.05	0.15
3	1.62	n	0.15	0.05	0.15
4	1.61*	n	0.15	0.05	0.15
5	1.6*	n	0.15	0.05	0.15
6	1.59	n	0.15	0.05	0.15
7	1.58*	n	0.15	0.05	0.15
8	1.57*	n	0.15	0.05	0.15
9	1.56*	n	0.15	0.05	0.15
10	1.55	n	0.15	0.05	0.15
11	1.5425*	n	0.15	0.05	0.15

Figure 6.38 Manning's n Data View and Editing Table

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.

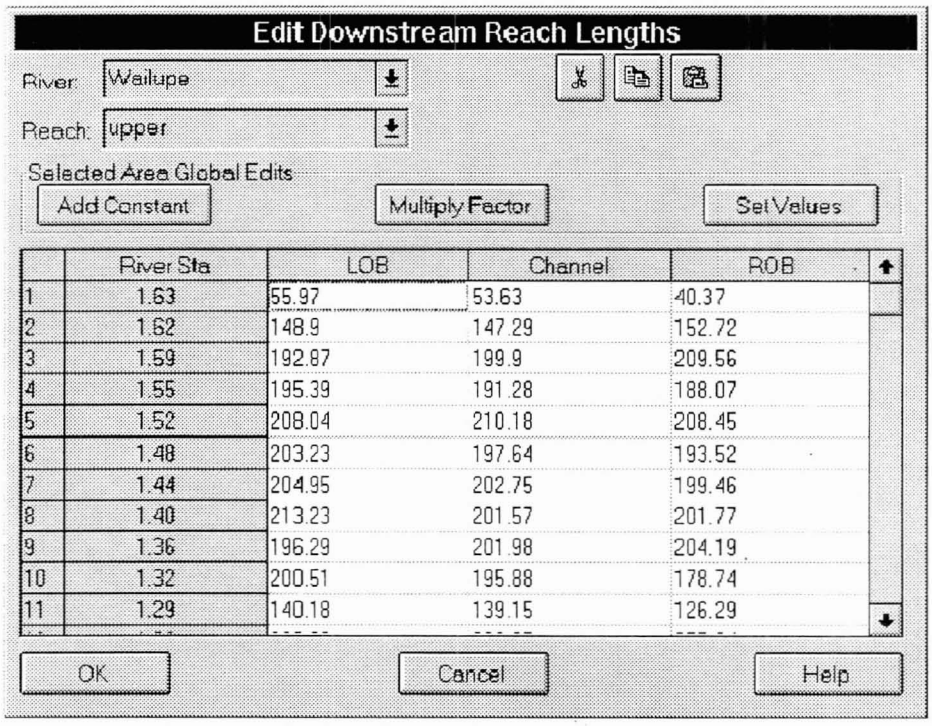
To multiply a group of n or k values by a factor, the user first highlights the desired cells. Once the cells are highlighted, pressing the **Multiply by a Factor** button will bring up a pop up window. This window allows the user to enter a value that will be multiplied by each of the highlighted cells.

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 6.39. 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.

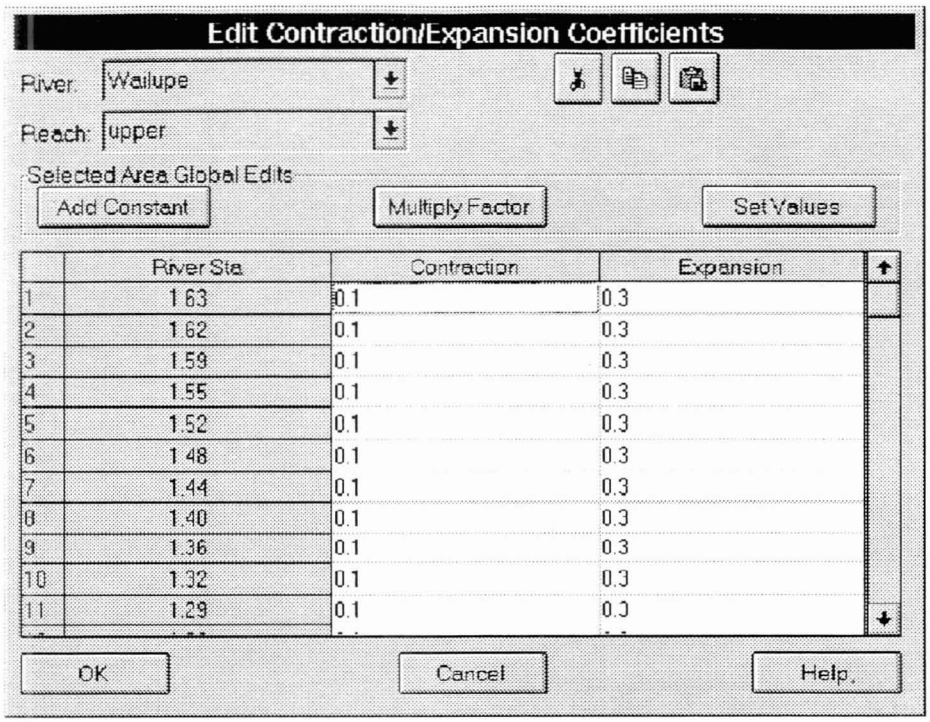


	River Sta	LOB	Channel	ROB
1	1.63	55.97	53.63	40.37
2	1.62	148.9	147.29	152.72
3	1.59	192.87	199.9	209.56
4	1.55	195.39	191.28	188.07
5	1.52	208.04	210.18	208.45
6	1.48	203.23	197.64	193.52
7	1.44	204.95	202.75	199.46
8	1.40	213.23	201.57	201.77
9	1.36	196.29	201.98	204.19
10	1.32	200.51	195.88	178.74
11	1.29	140.18	139.15	126.29

Figure 6.39 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 6.40. 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	1.63	0.1	0.3
2	1.62	0.1	0.3
3	1.59	0.1	0.3
4	1.55	0.1	0.3
5	1.52	0.1	0.3
6	1.48	0.1	0.3
7	1.44	0.1	0.3
8	1.40	0.1	0.3
9	1.36	0.1	0.3
10	1.32	0.1	0.3
11	1.29	0.1	0.3

Figure 6.40 Contraction and Expansion Coefficients Table

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 6.41. 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.

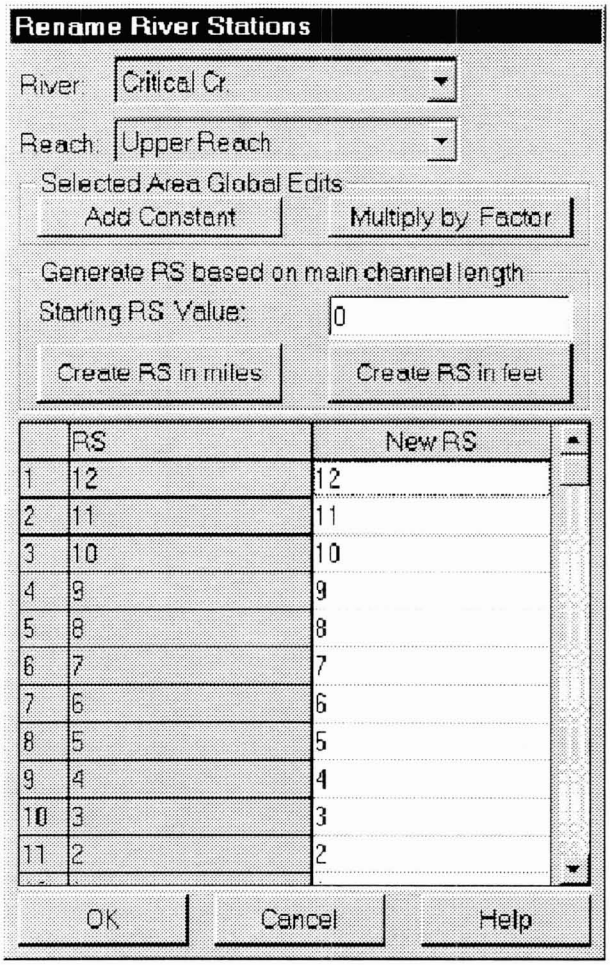


Figure 6.41 Cross Section River Stationing View and Editing Table

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; the HEC-2 data format, and the HEC-RAS data format. Data can be imported into an existing HEC-RAS geometry file or for a completely new geometry file (one that has no data at all). Multiple data files can be imported into the same geometric data file on a reach by reach basis.

To import data into an 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 13 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.

Graphical Cross Section Editor

A graphical cross section editor is available from the **Options** menu of the Geometric Data Editor window. When this option is selected, a window will appear as shown in Figure 6.42.

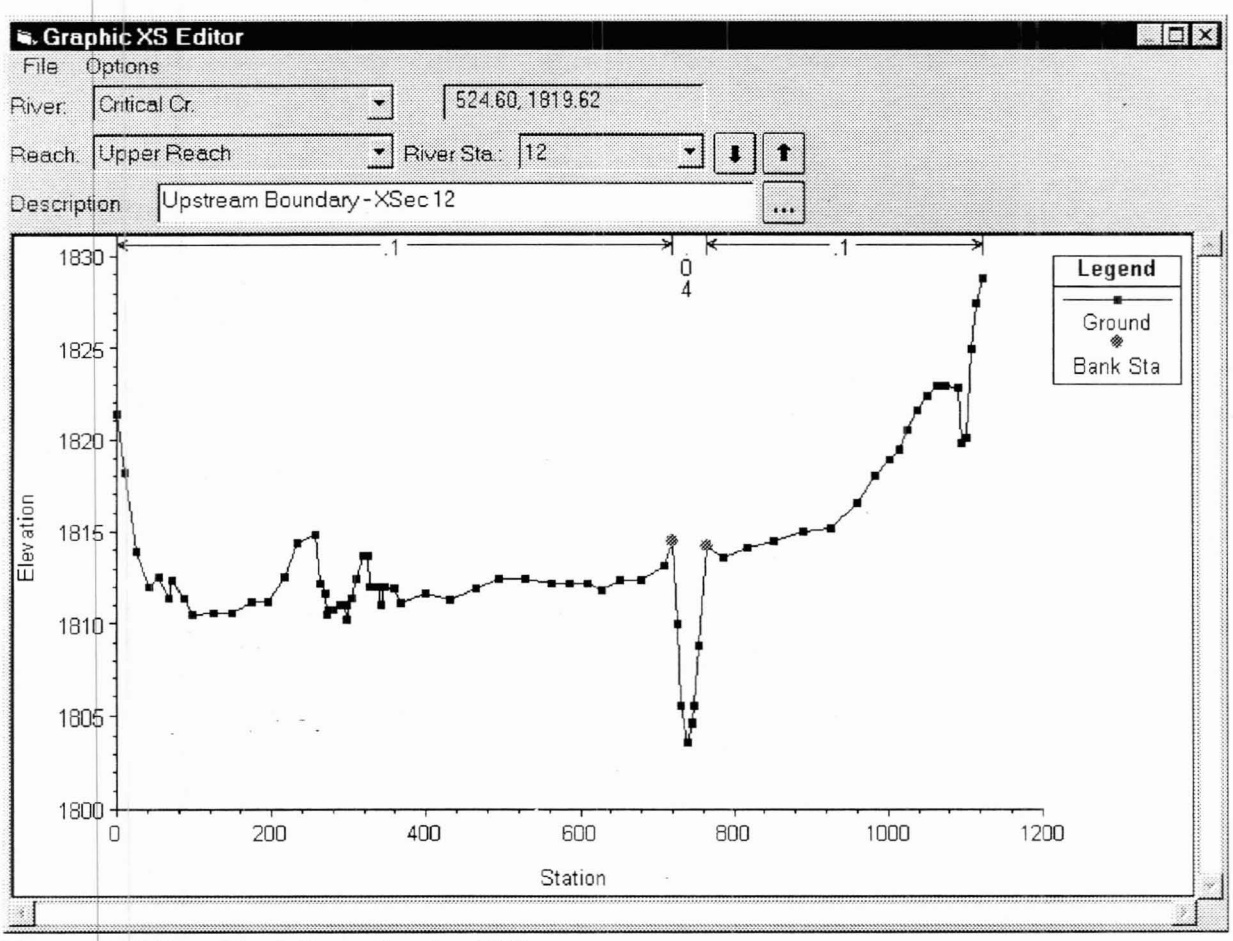


Figure 6.42Graphical Cross Section Editor

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

Other available options from the Graphical Cross Section editor are the ability to zoom in and zoom out, to overlay a grid onto the cross section plot, and to undo all of the graphical editing. When the **Undo Edits** option is selected, the cross section is automatically returned to its original state before this particular editing session began. However, once this editor is closed, or if the user selects a different cross section from the editor, it is assumed that the user is happy with the changes that were made and they are saved in memory. The data is not saved to the hard disk, so it is still possible to get the original data back if needed.

Attaching and Viewing Pictures

The user can attach a picture to any cross section or hydraulic structure (bridge, culvert, etc.). Once pictures are attached, they can be viewed from a picture viewer within the HEC-RAS geometric data editor. The picture viewer supports the following graphics formats: bit map (*.bmp); icon (*.ico); windows metafile (*.wmf); GIF (*.gif); and JPEG (*.jpg).

Pictures are attached to cross sections or hydraulic structures from within the picture viewer. To bring up the picture viewer, go to the geometric data editor and click on the **View Picture** button with the left mouse button. An editor will appear as shown in Figure 6.43. To attach a picture to a particular river station, first select the River, Reach, and River Station in which you would like to attach the picture. Next select the **Add Picture** button, and a file selection box will appear allowing you to select a graphics file to attach to the selected location. If the picture file is not in the same location as your data files, you can select the drive and path of the picture from within the file

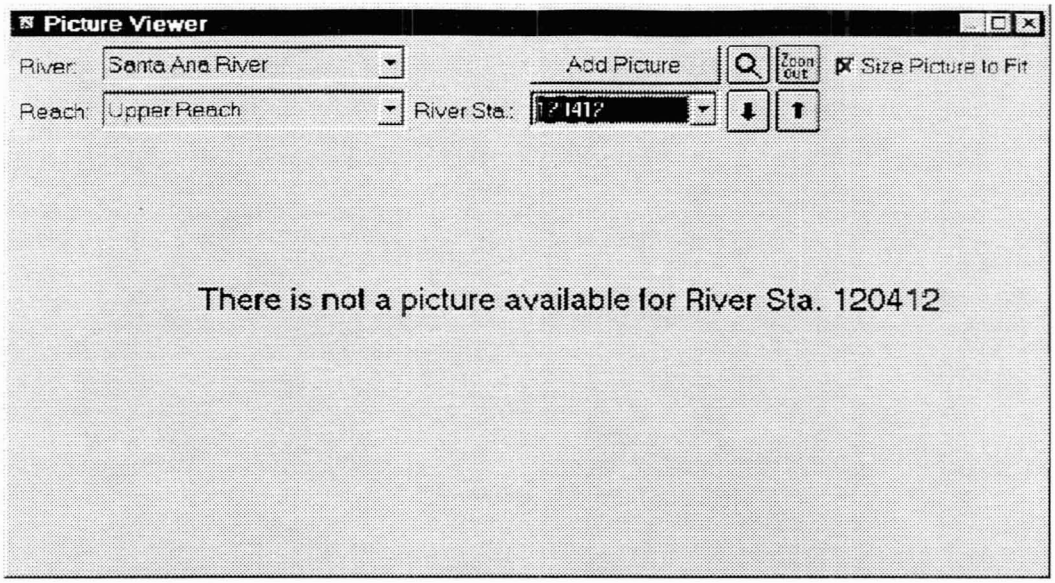


Figure 6.43 HEC-RAS Picture Viewer

selection box. Once a graphic file is located and selected, press the **Open** button to attach it to the selected location. The picture should automatically show up inside of the picture viewer. An example picture is shown in Figure 6.44. Additional pictures can be added by selecting a different location and selecting the Add Picture button to attach the picture. Only one picture can be attached to a model object.

Once pictures are attached to the viewer, the user can move to different pictures by using the up and down arrow buttons, or selecting a specific river stationing that has a picture attached to it. Options are available to zoom in, zoom out, remove pictures, and to resize the picture to fit within the size of the picture viewer window. The user can resize the picture viewer to whatever size they want. However, if you are viewing a bitmap picture, and you make the window larger than the actual picture resolution, the photo will begin to distort.

Once pictures are attached to the geometry file, a small red square will be displayed on the river system schematic at each location where a picture exists. When the user clicks the left mouse button over a cross section, a pop up menu will appear. If that particular cross section has a picture attached to it, one of the menu options will be to view the picture. Selecting the **View Picture** option from the pop up menu will bring up the picture viewer and automatically load that particular picture.

The pictures are stored as part of the geometry data (not the actual picture, but its location on the hard disk). In general, it is a good idea to keep the picture files in the same directory as your project data files. This will make it easier to keep track of all the files associated with a particular project.

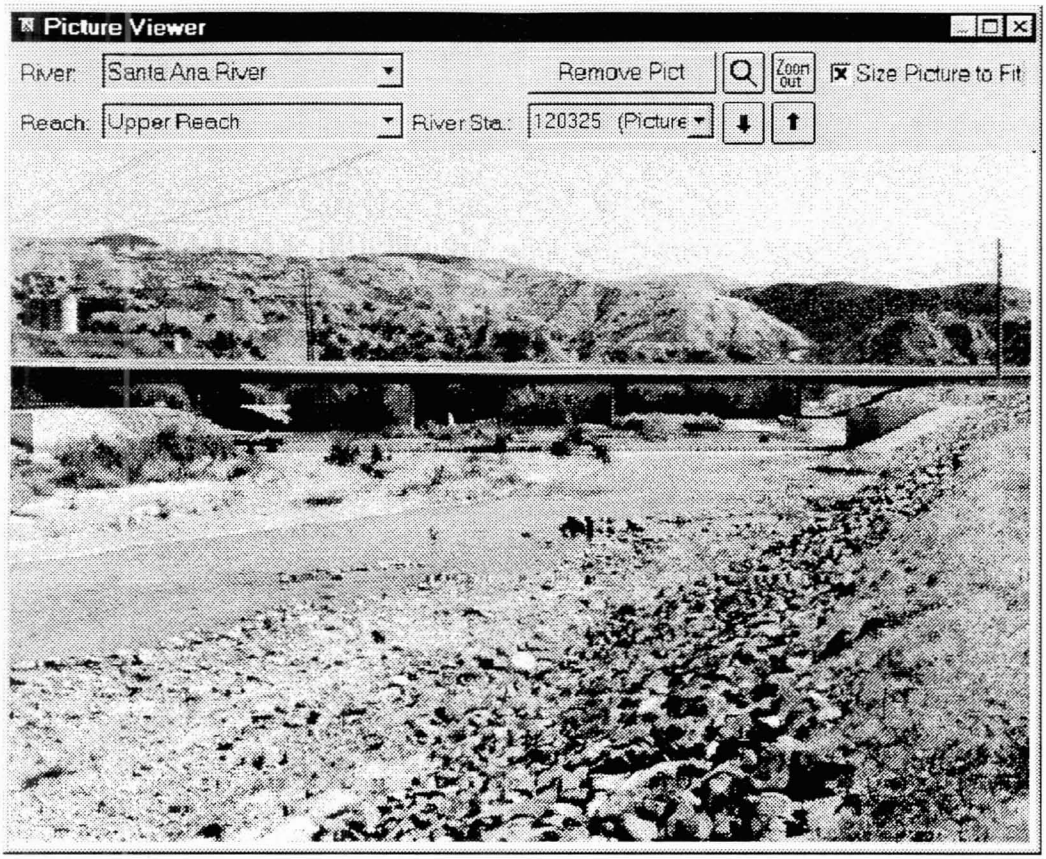


Figure 6.44 Picture Viewer with Example Bit Map Photo

Saving the Geometric Data

To save the geometric data, use the Save Geometry Data As option from the File menu of the Geometric Data window. When this option is selected, the user is prompted to enter a title for the geometric data. Once you have entered the title, press the OK button and the data will be saved to the hard disk. If the geometric data have been saved before (and therefore a title has already been entered), then it is only necessary to select the Save Geometry Data option. When this option is selected, the geometry data are saved with the previously defined title.

In general, it is a good idea to periodically save your data as you are entering them. This will prevent the loss of large amounts of information in the event of a power failure, or if a program error occurs in the HEC-RAS user interface.

This Page Intentionally Left Blank

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

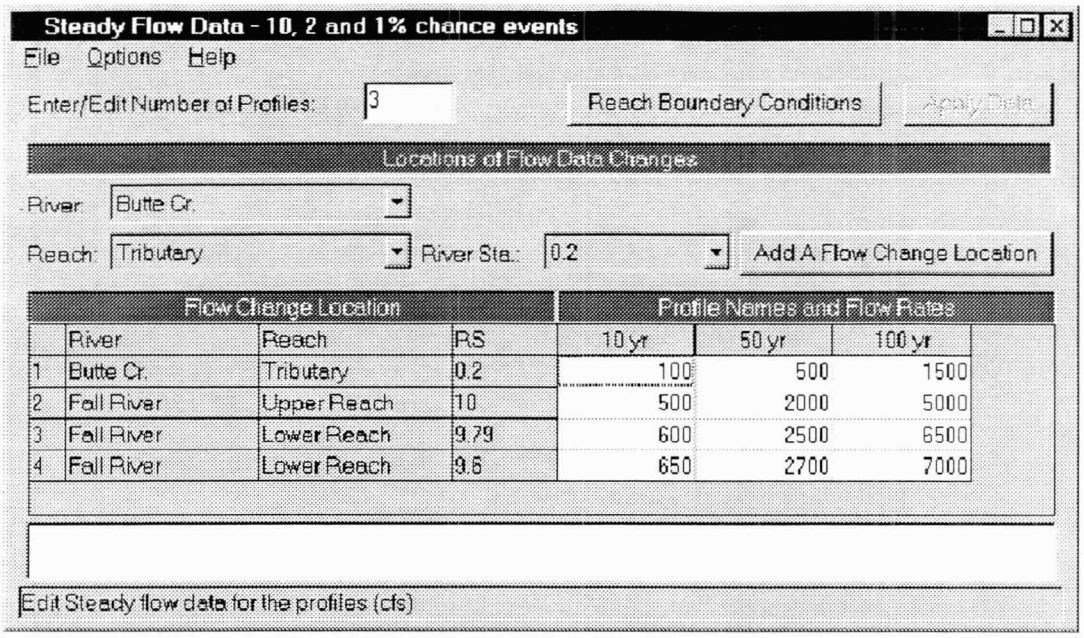
Entering and Editing Steady Flow Data

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 flow 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.

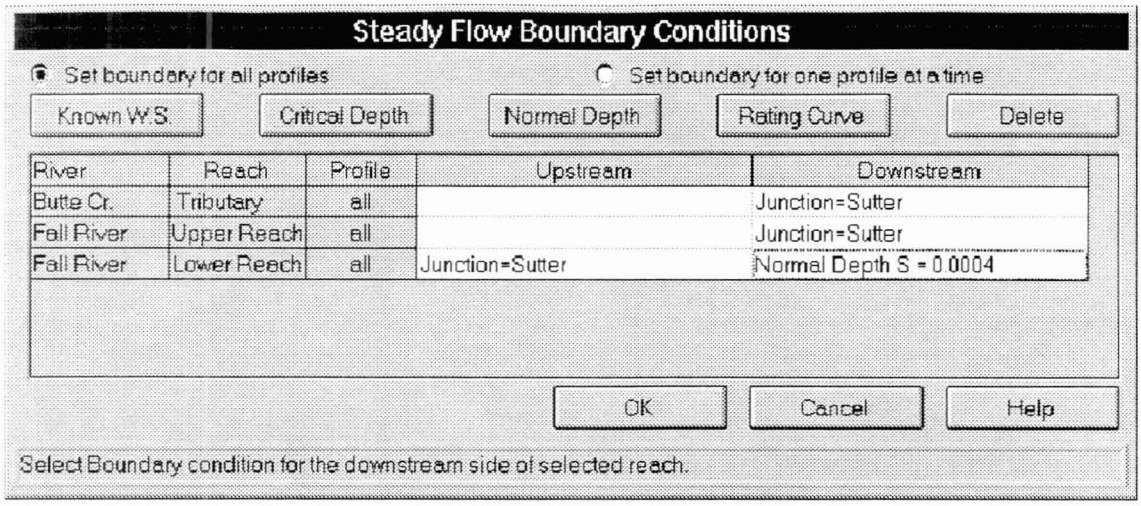


Figure 7.2 Steady Flow Boundary Conditions Editor

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 data, 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.

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.

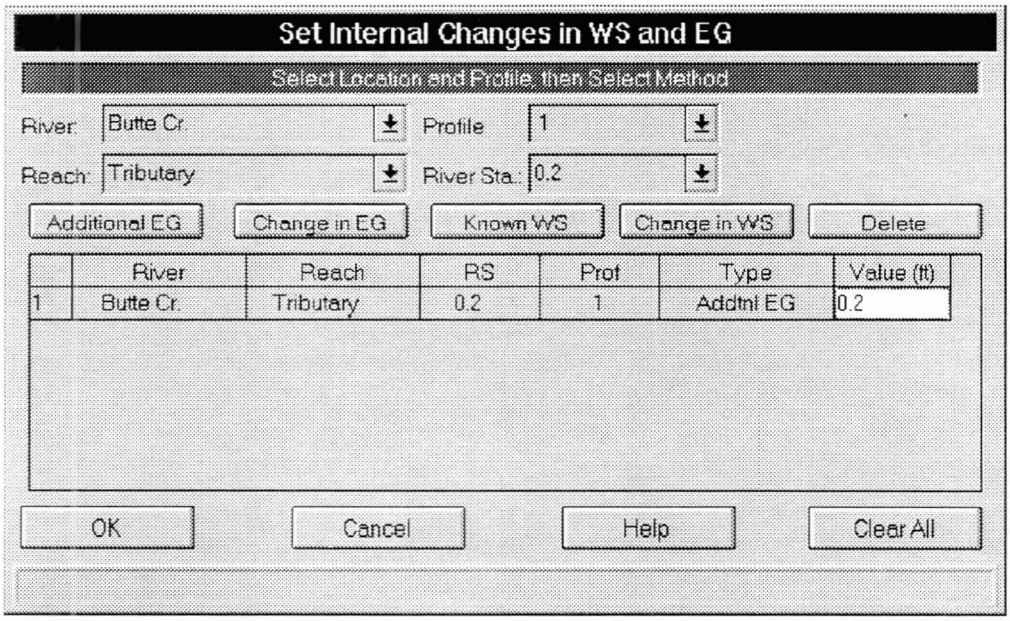
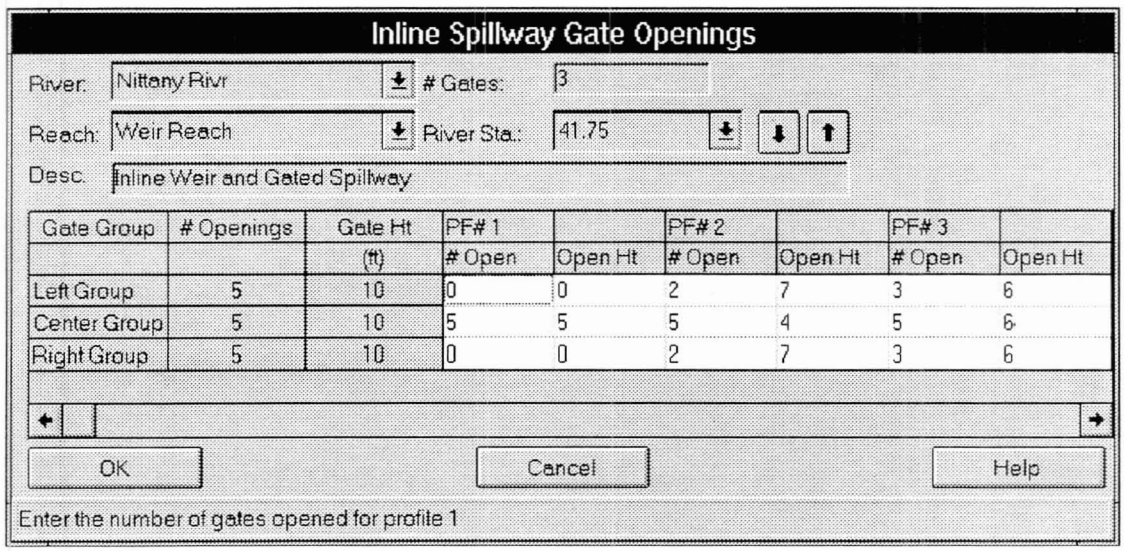


Figure 7.3 Setting Changes in Water Surface and Energy

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.

Inline Spillway Gate Openings. This option allows the user to control gate openings for any inline 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.



Gate Group	# Openings	Gate Ht	PF# 1		PF# 2		PF# 3	
		(ft)	# Open	Open Ht	# Open	Open Ht	# Open	Open Ht
Left Group	5	10	0	0	2	7	3	6
Center Group	5	10	5	5	5	4	5	6
Right Group	5	10	0	0	2	7	3	6

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.

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 80 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.

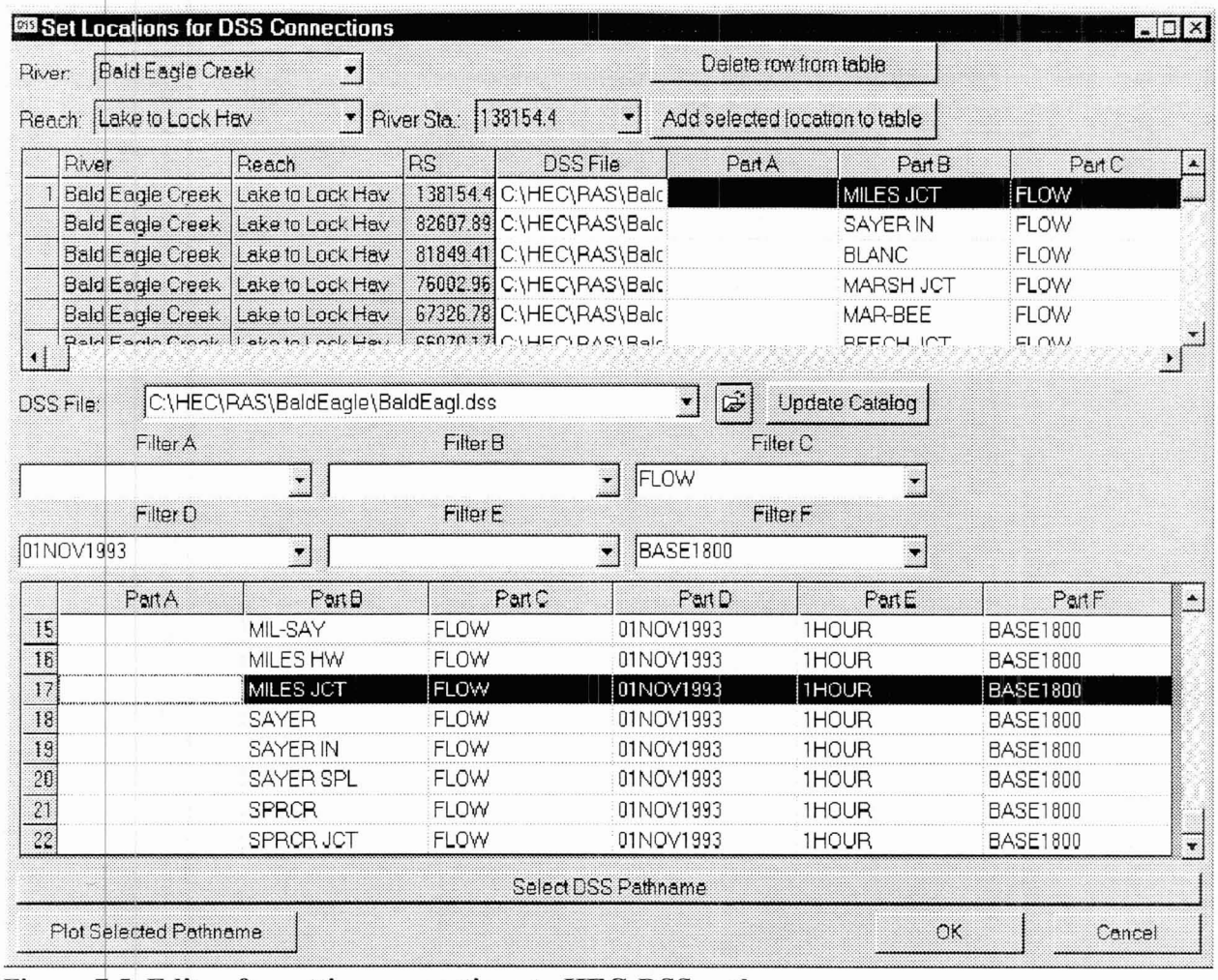


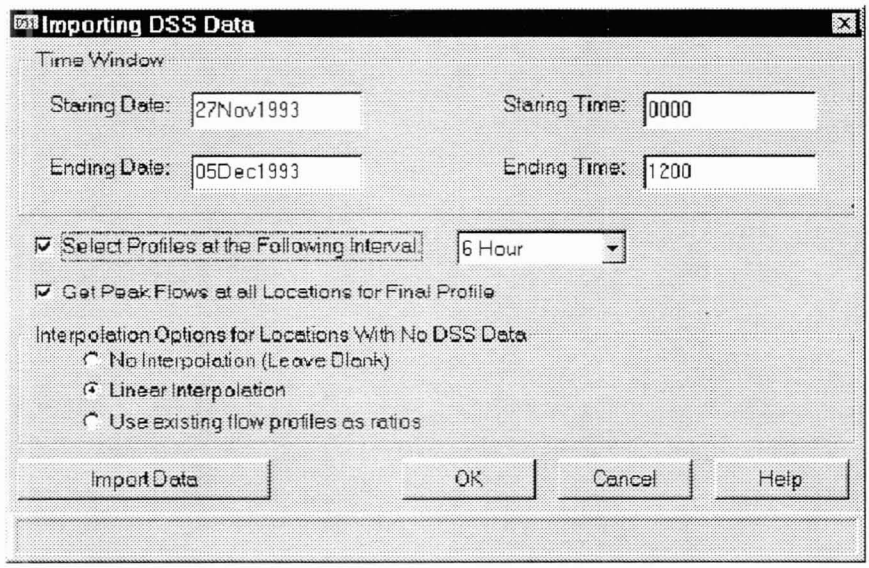
Figure 7.5 Editor for setting connections to HEC-DSS pathnames.

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 window. 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 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



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 **Simulate** menu. The Steady Flow Analysis window will appear as in Figure 7.7 (except yours may not have a Plan title and short ID).

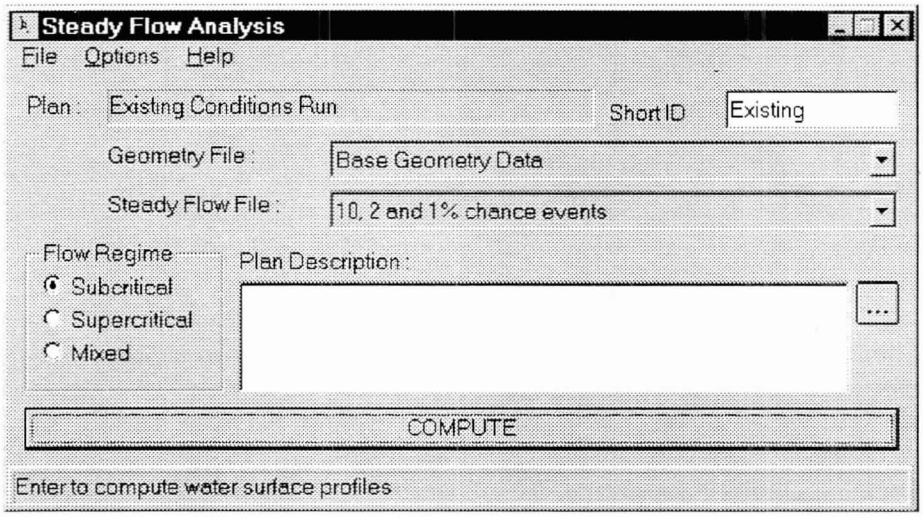


Figure 7.7 Steady Flow Analysis window

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 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 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.

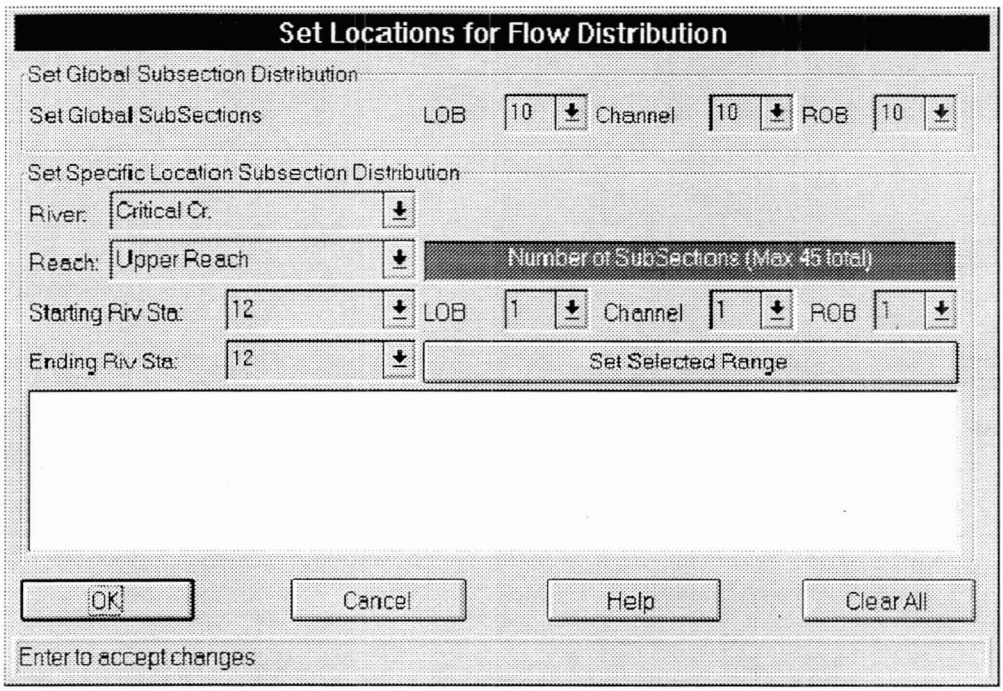


Figure 7.8 Window for Specifying the Locations of Flow Distribution.

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 8 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 guesses 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.

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.

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 7.9. 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.

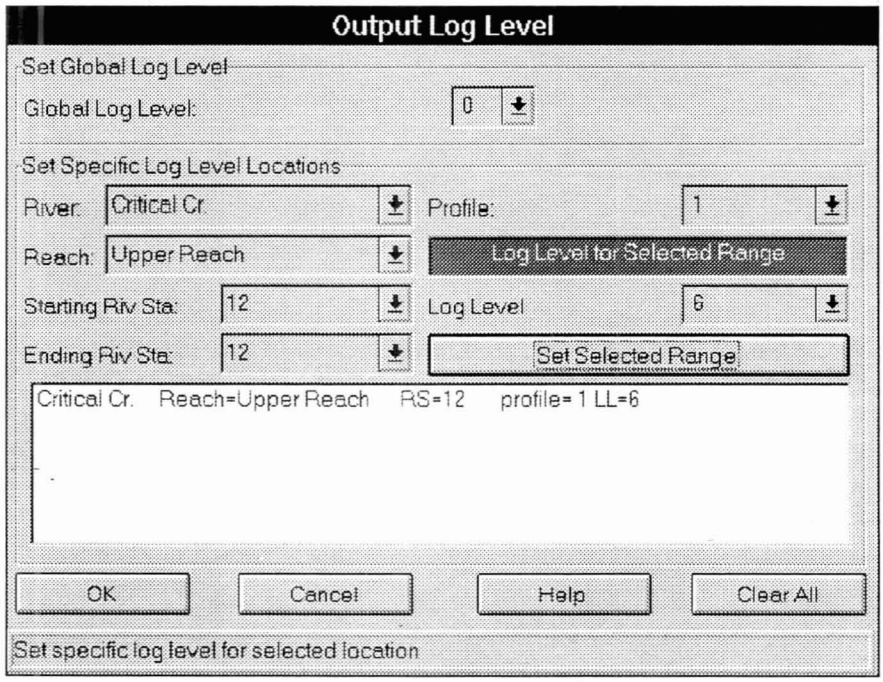


Figure 7.9 Log File Output Level window

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.

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 double clicking the upper left corner of the window. If the computations ended with a message stating "**PROGRAM TERMINATED NORMALLY**," the user can then begin to review the output.

CHAPTER 8

Viewing Results

After the model has finished the steady flow profile computations, you 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, X-Y-Z perspective plots, detailed tabular output at a single cross section, 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
- X-Y-Z Perspective Plots
- Tabular Output
- View Results from the River System Schematic
- Viewing Flow Distribution Output

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 either **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 8.1. 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 arrow buttons next to the river station box.

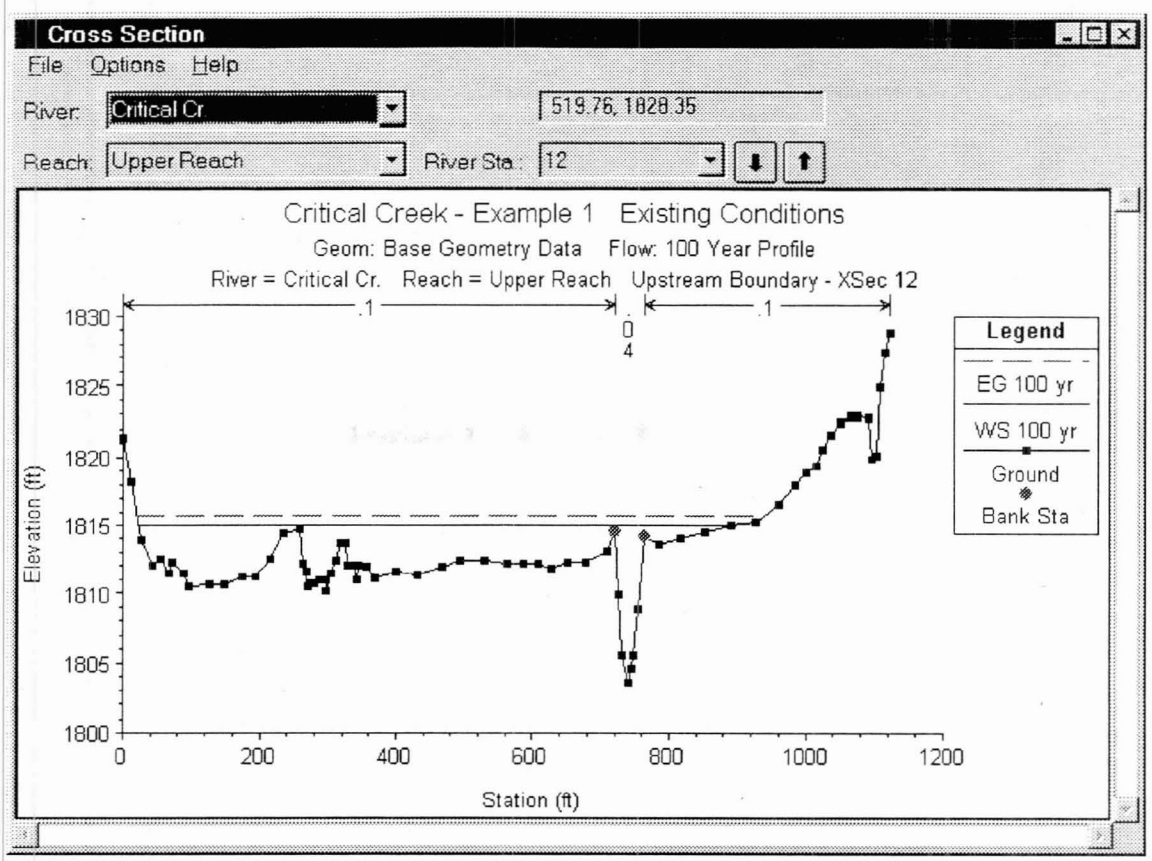


Figure 8.1 Example Cross Section Plot

An example profile plot is shown in Figure 8.2. 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 the plot.

An example rating curve plot is shown in Figure 8.3. 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.

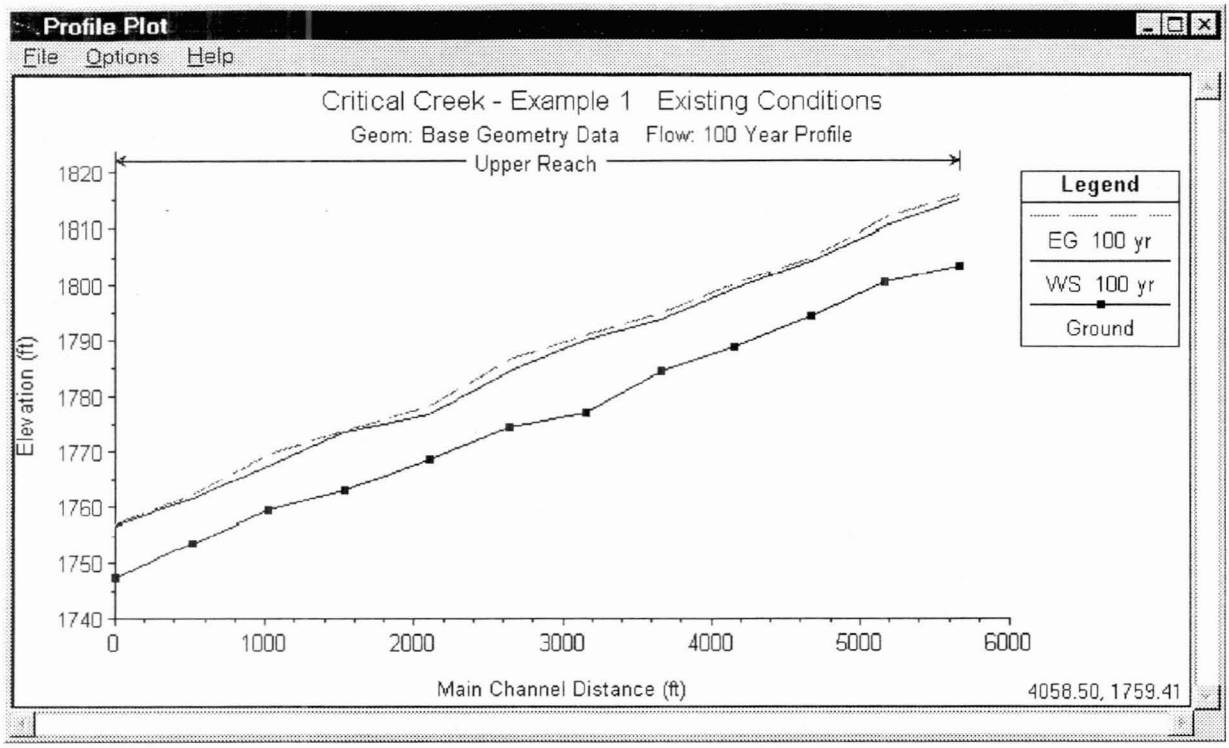


Figure 8.2 Example Profile Plot

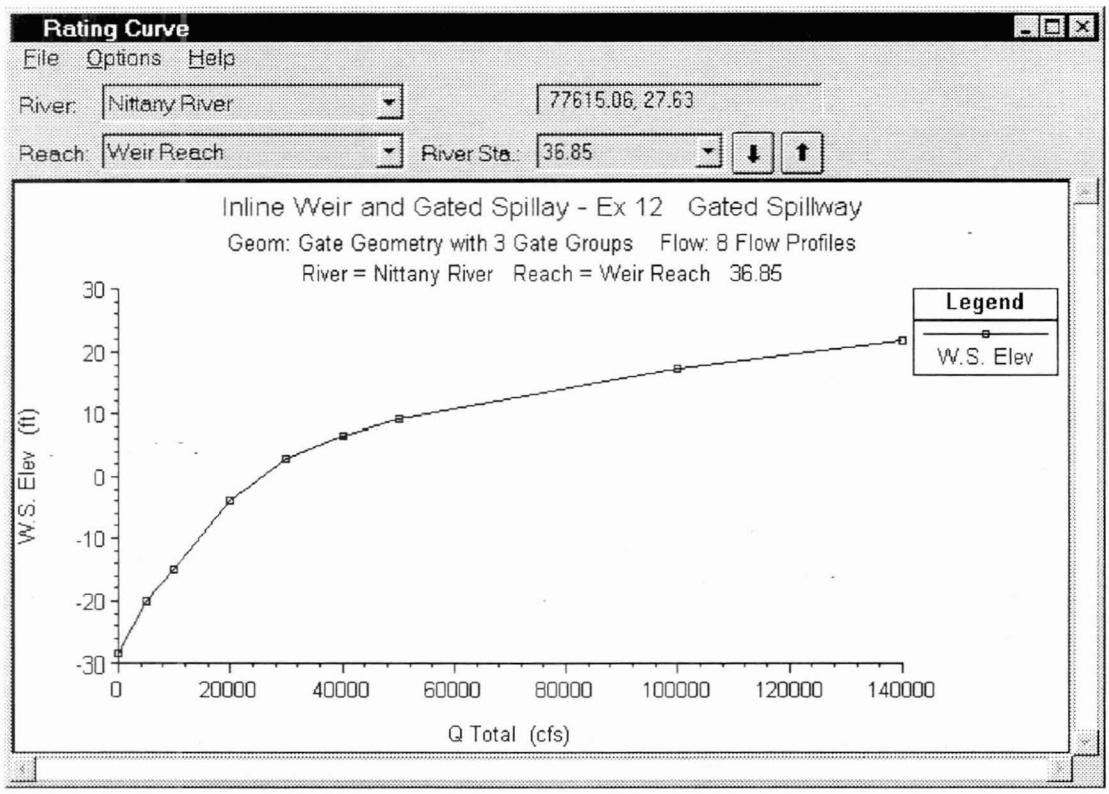


Figure 8.3 Example Rating Curve Plot

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.

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 what ever 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. The cross section plot is limited to the following four variables: water surface, energy, critical water surface, and Manning's n values.

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 8.4.

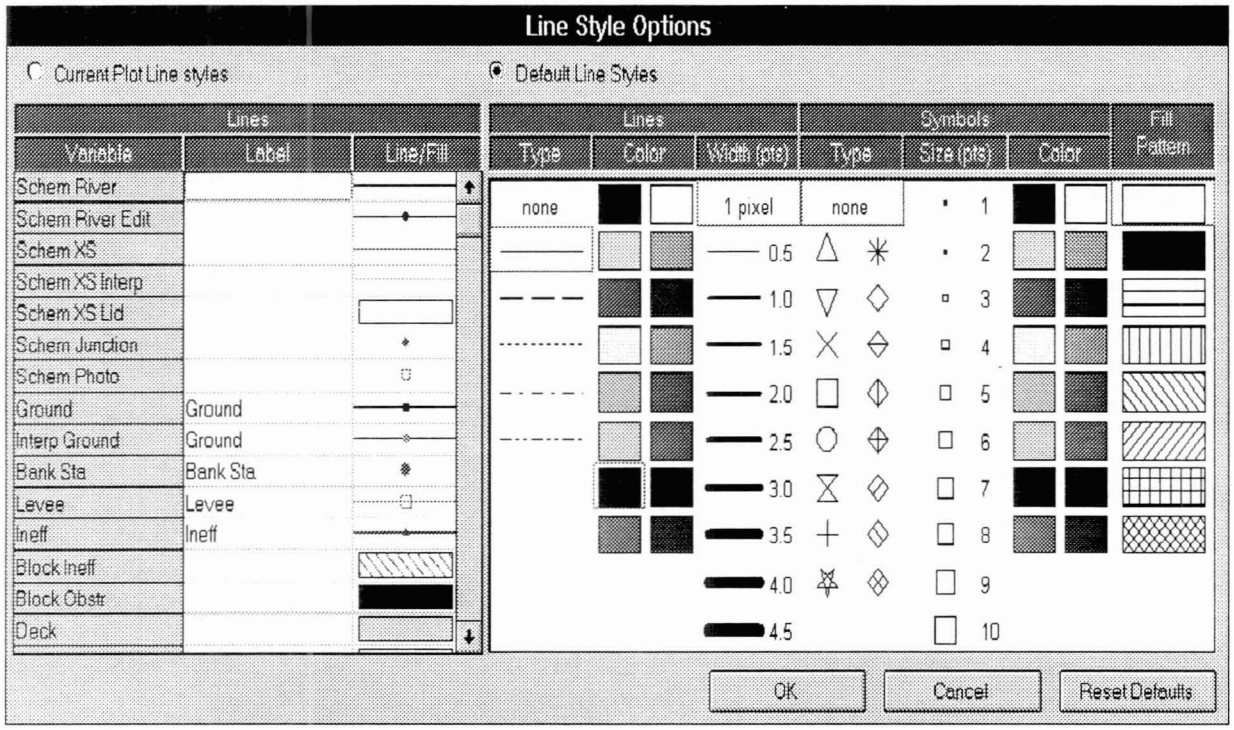


Figure 8.4 Line and Symbol Options Window

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 can not 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 zoom window. The zoom window appears whenever the user has selected the Zoom In option.

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 8.5) 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 8.6.

For details on how to select the locations for computing the velocity distribution, see Chapter 7 of the User's Manual. For information on how the velocity distribution is actually calculated, see Chapter 4 of the Hydraulic Reference Manual.

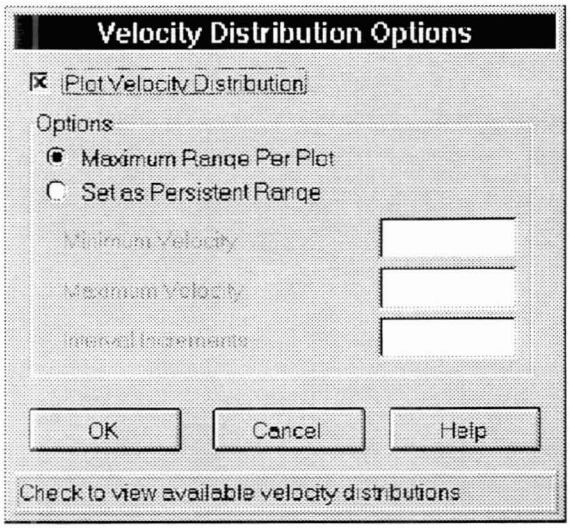


Figure 8.5 Velocity Distribution Options.

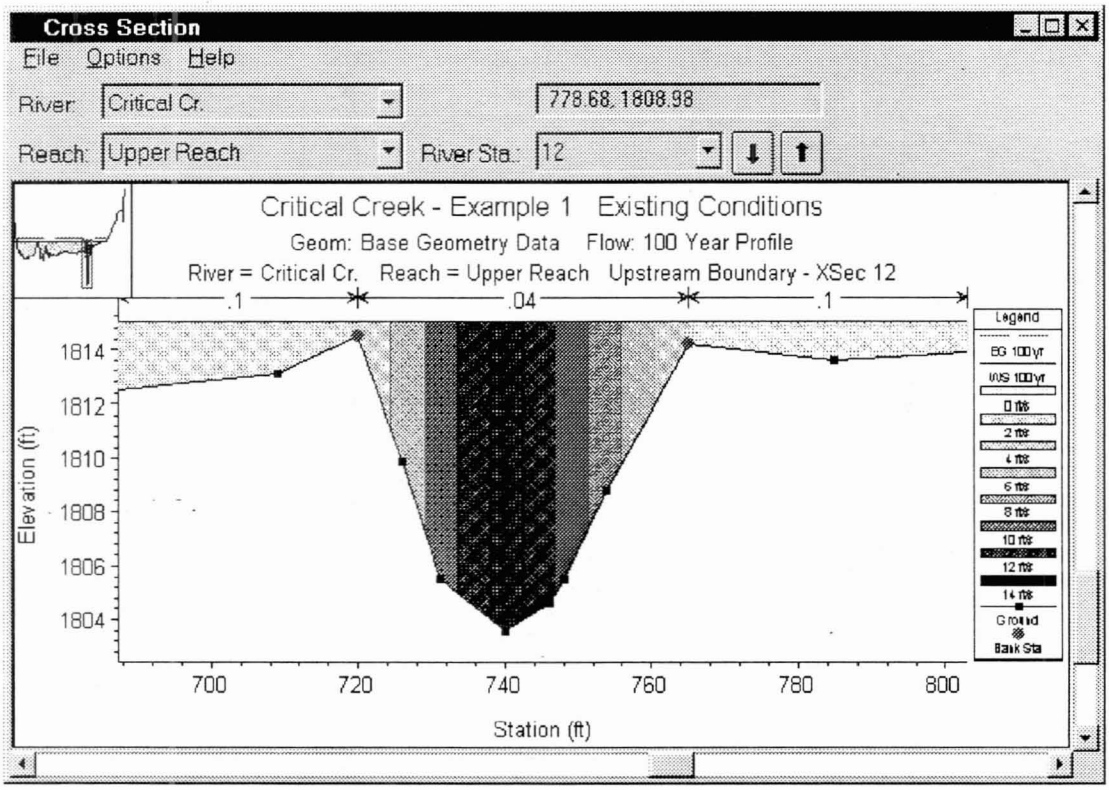


Figure 8.6 Velocity Distribution Plot

Plotting Other Variables in Profile

The profile plotting window has the ability to plot other variables, besides water surface and energy, in profile. Any variable that is computed at a cross section can be displayed in profile. An example would be to plot velocity versus distance. To plot other variables in profile, the user must first select the **General Profile** option from the **Options** menu of the Profile plot. Once this option is turned on, the plot automatically switches to a plot of channel velocity versus distance. Other variables can be selected from the **Variables** option under the **Options** menu. 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 8.7.

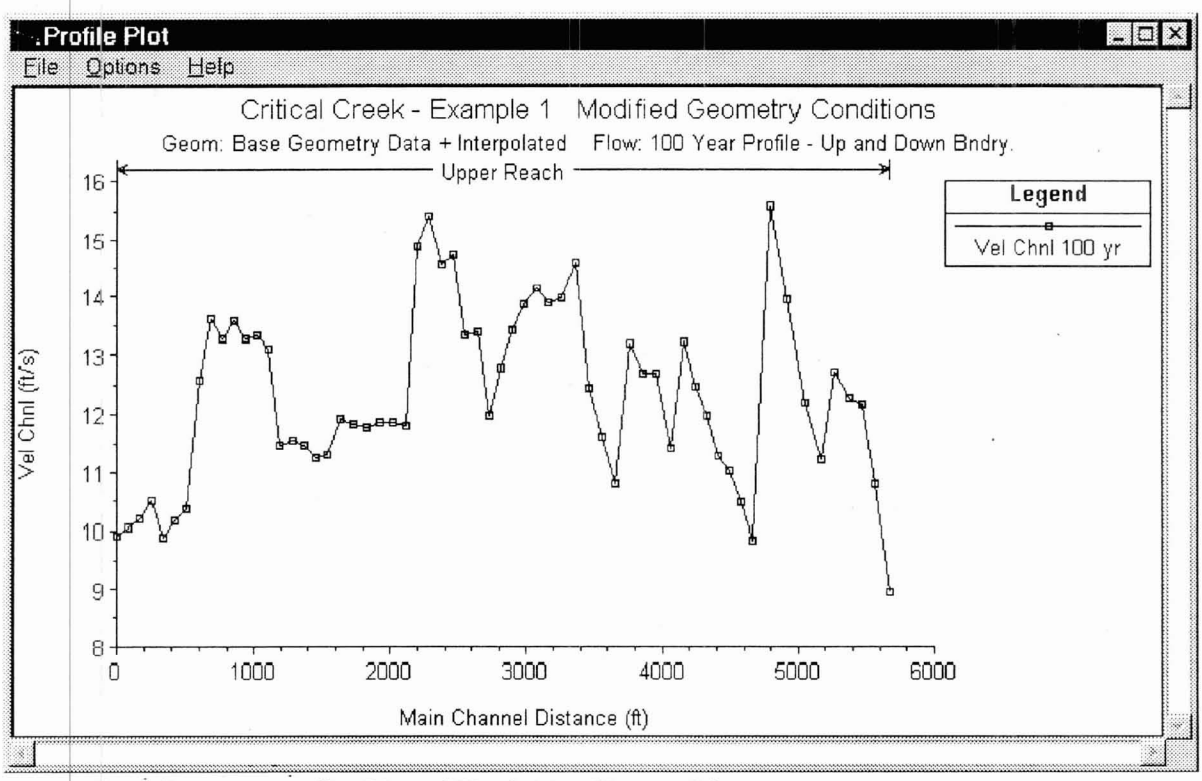


Figure 8.7 General Profile Plot of Variables Versus Distance

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 8.8. 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.

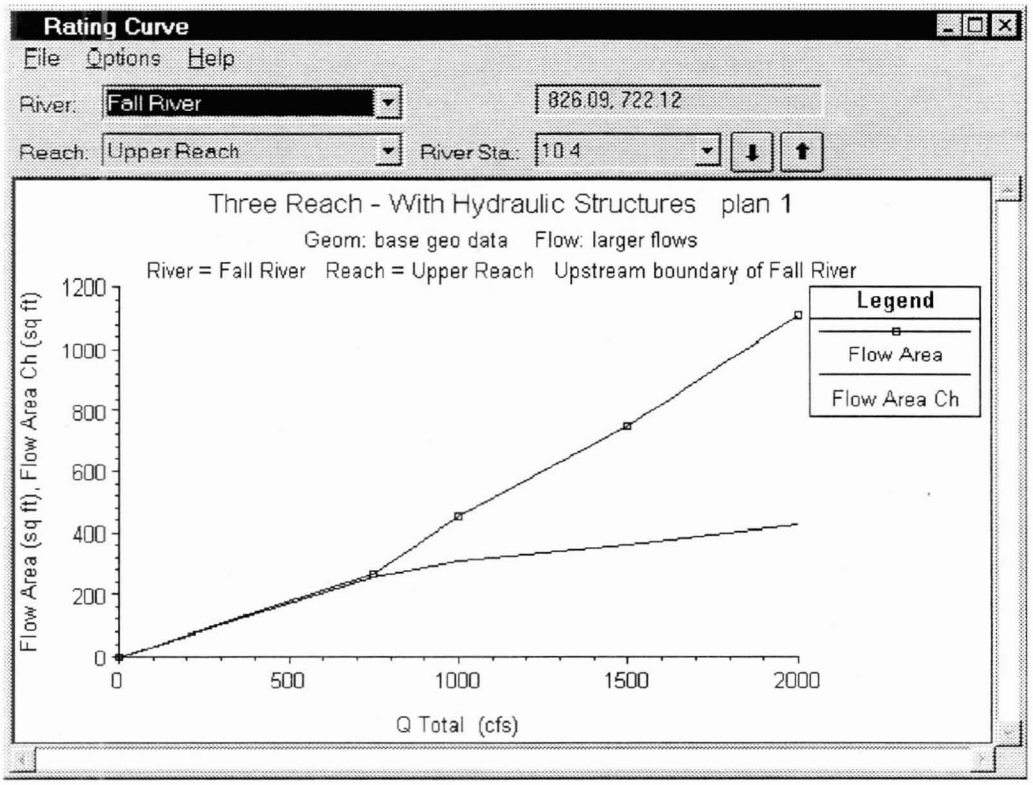


Figure 8.8 Example of Plotting One Variable Against Other Variables

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.

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 8.9.

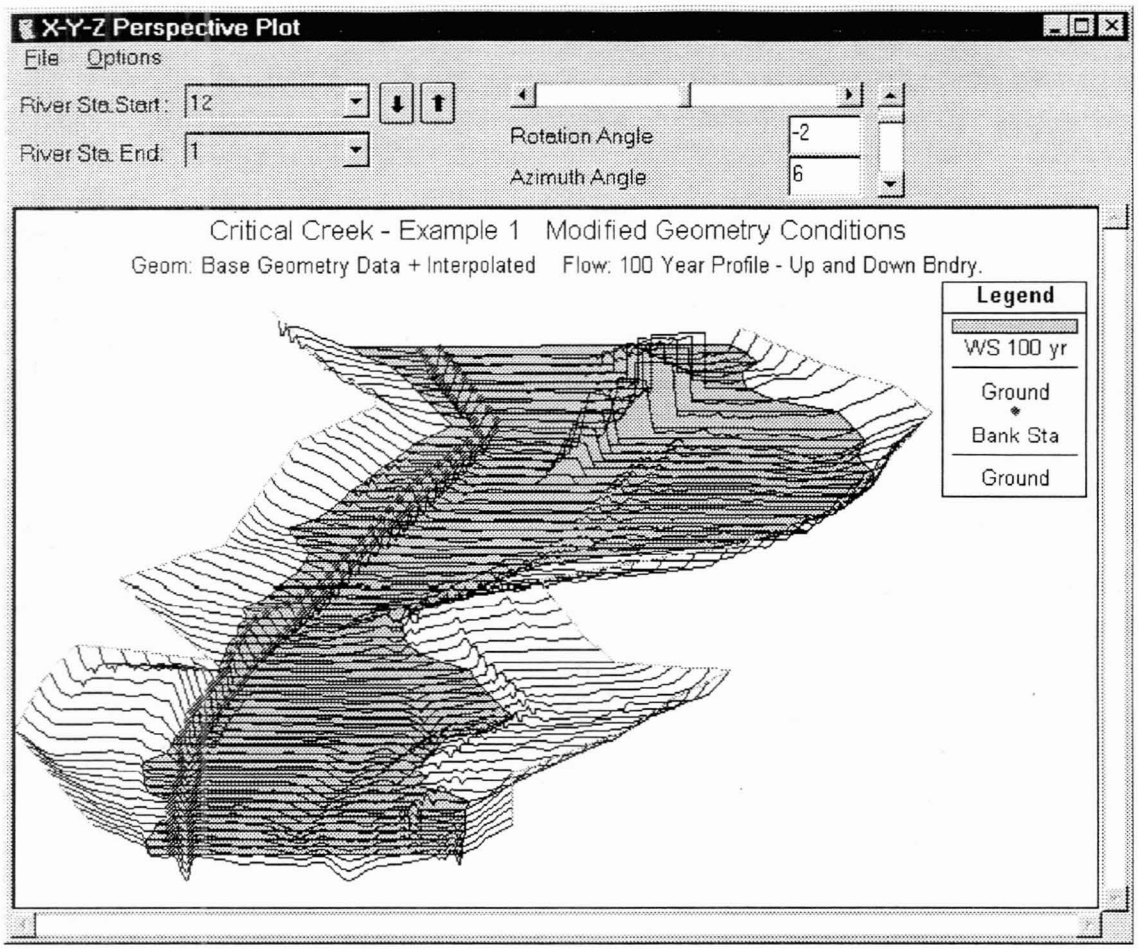


Figure 8.9 Example X-Y-Z Perspective Plot

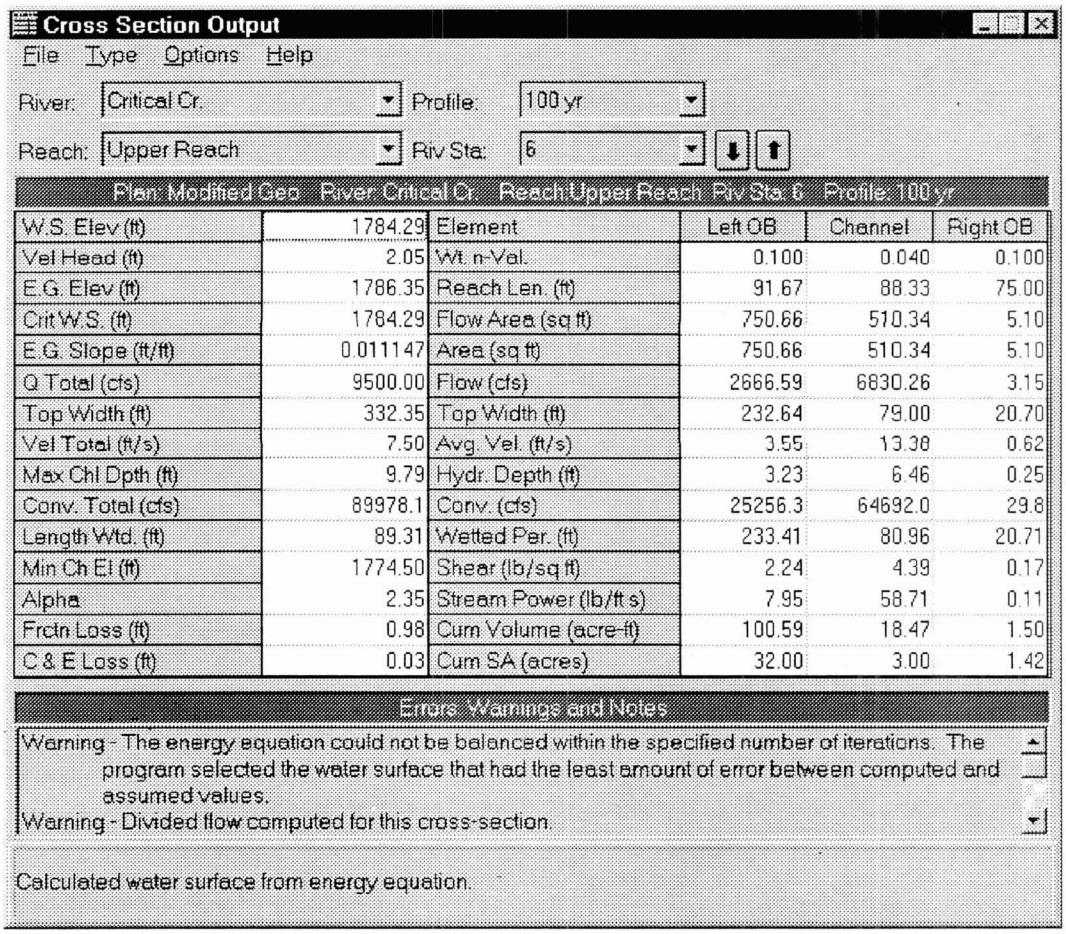
The user has the ability to select which reaches to be plotted, the range of the river stations, and which plans and profiles to be displayed. The plot can be rotated left and right, as well as up and down, in order to get different perspectives of the river system. Zoom in and zoom out features are 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, cross section tables and profile tables.

Cross Section Tables

Cross section tables show detailed hydraulic information at a single location, for a single profile. To display a cross section table on the screen, select **Cross Section Table** from the **View** menu of the main HEC-RAS window. An example cross section table is shown in Figure 8.10.



		Element	Left OB	Channel	Right OB
W.S. Elev (ft)	1784.29				
Vel Head (ft)	2.05	Wt. n-Val.	0.100	0.040	0.100
E.G. Elev (ft)	1786.35	Reach Len. (ft)	91.67	88.33	75.00
Crit W.S. (ft)	1784.29	Flow Area (sq ft)	750.66	510.34	5.10
E.G. Slope (ft/ft)	0.011147	Area (sq ft)	750.66	510.34	5.10
Q Total (cfs)	9500.00	Flow (cfs)	2666.59	6830.26	3.15
Top Width (ft)	332.35	Top Width (ft)	232.64	79.00	20.70
Vel Total (ft/s)	7.50	Avg. Vel. (ft/s)	3.55	13.38	0.62
Max Chl Dpth (ft)	9.79	Hydr. Depth (ft)	3.23	6.46	0.25
Conv. Total (cfs)	89978.1	Conv. (cfs)	25256.3	64692.0	29.8
Length Wtd. (ft)	89.31	Wetted Per. (ft)	233.41	80.96	20.71
Min Ch El (ft)	1774.50	Shear (lb/sq ft)	2.24	4.39	0.17
Alpha	2.35	Stream Power (lb/ft s)	7.95	58.71	0.11
Frctn Loss (ft)	0.98	Cum Volume (acre-ft)	100.59	18.47	1.50
C & E Loss (ft)	0.03	Cum SA (acres)	32.00	3.00	1.42

Figure 8.10 Example Cross Section Table

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.

Users can also view detailed hydraulic information for other types of cross section specific tables. Other table types are selected from the **Type** menu on the cross section 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 8.11.

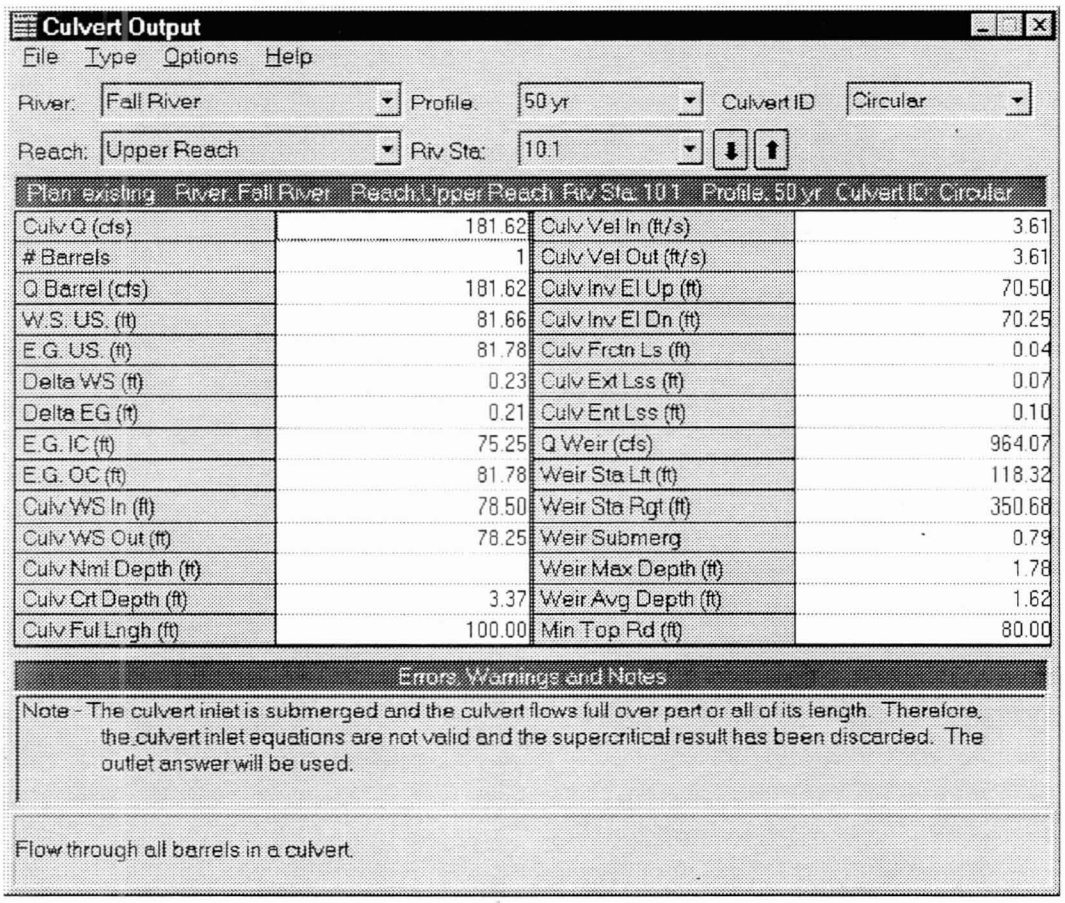


Figure 8.11 Example Culvert Type of Cross Section Table

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 8.12.

Bridge Output

File Type Options Help

River: Butte Creek Profile: 50 yr Opening Bridge #1

Reach: Tributary Riv Sta: 0.22

Plan: existing River: Butte Creek Reach: Tributary Riv Sta: 0.22 Profile: 50 yr Opening: Bridge #1

			Inside BR US	Inside BR DS
E.G. US. (ft)	82.90	Element		
W.S. US. (ft)	82.77	E.G. Elev (ft)	82.90	82.88
Q Total (cfs)	500.00	W.S. Elev (ft)	82.77	82.59
Q Bridge (cfs)	339.75	Crit W.S. (ft)	76.73	76.72
Q Weir (cfs)	160.25	Max Chl Dpth (ft)	12.77	12.59
Weir Sta Lft (ft)	218.88	Vel Total (ft/s)	4.10	4.64
Weir Sta Rgt (ft)	298.75	Flow Area (sq ft)	121.97	107.79
Weir Submerg	0.27	Froude # Chl	0.24	0.27
Weir Max Depth (ft)	0.90	Specif Force (cu ft)	578.31	567.12
Min Top Rd (ft)	82.00	Hydr Depth (ft)	1.56	1.43
Min El Prs (ft)	79.00	W.P. Total (ft)	130.98	128.37
Delta EG (ft)	0.48	Conv. Total (cfs)		
Delta WS (ft)	0.52	Top Width (ft)	78.13	75.60
BR Open Area (sq ft)	65.66	Frctn Loss (ft)		
BR Open Vel (ft/s)	5.17	C & E Loss (ft)		
Coef of Q		Shear Total (lb/sq ft)		
Br Sel Mthd	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.

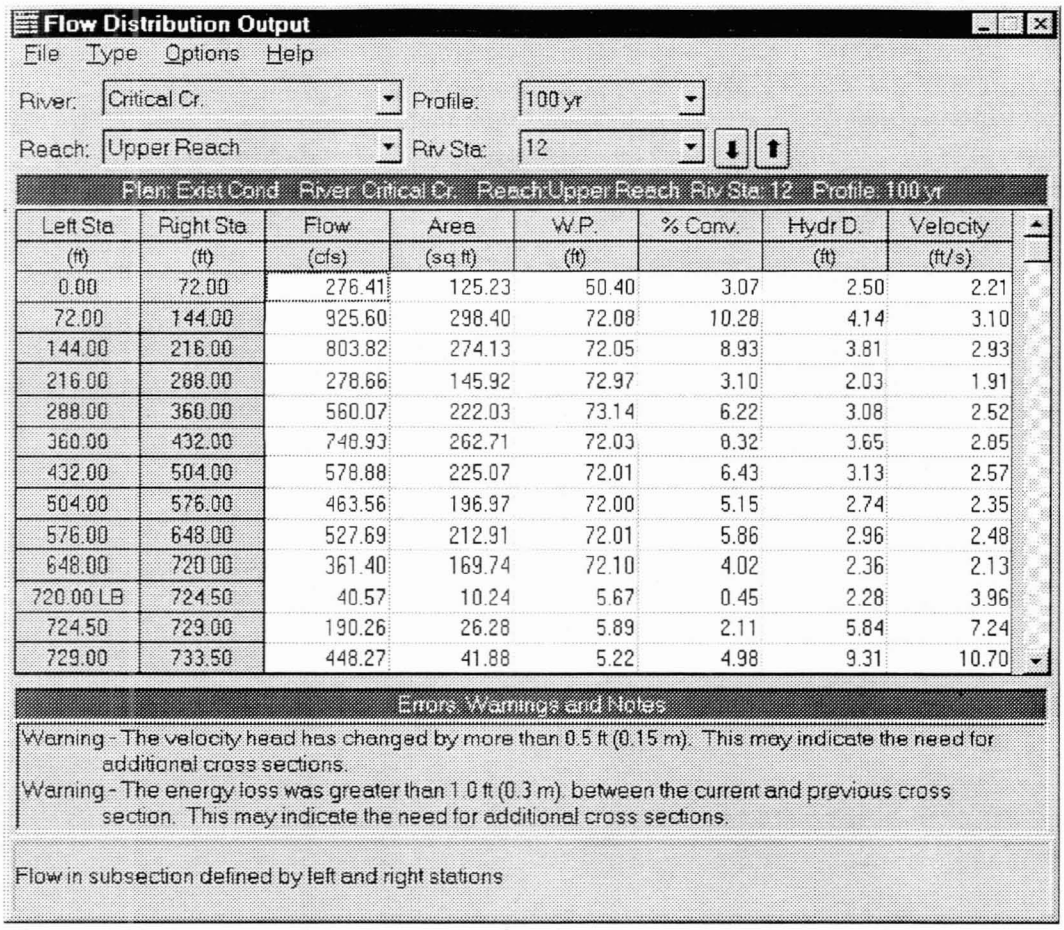
Average velocity inside the bridge opening (Maximum of BU and BD)

Figure 8.12 Example Bridge Type of Cross Section Table

Conveyance. The conveyance type of table is the same as a normal cross section table. The only difference is that a conveyance type of table is used to view the hydraulic results of a portion of a multiple opening river crossing. That is, if the user has defined an open channel flow type of opening (i.e., conveyance area), at a multiple opening crossing, then this table can be used to view the hydraulic results for that specific opening.

Inline Weir/Spillway. The Inline Weir/Spillway 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.

Flow Distribution. 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 8.13.



Flow Distribution Output

File Type Options Help

River: Critical Cr. Profile: 100 yr

Reach: Upper Reach Riv Sta: 12

Plan: Exist Cond River: Critical Cr. Reach: Upper Reach Riv Sta: 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	276.41	125.23	50.40	3.07	2.50	2.21
72.00	144.00	925.60	298.40	72.08	10.28	4.14	3.10
144.00	216.00	803.82	274.13	72.05	8.93	3.81	2.93
216.00	288.00	278.66	145.92	72.97	3.10	2.03	1.91
288.00	360.00	560.07	222.03	73.14	6.22	3.08	2.52
360.00	432.00	748.93	262.71	72.03	8.32	3.65	2.85
432.00	504.00	578.88	225.07	72.01	6.43	3.13	2.57
504.00	576.00	463.56	196.97	72.00	5.15	2.74	2.35
576.00	648.00	527.69	212.91	72.01	5.86	2.96	2.48
648.00	720.00	361.40	169.74	72.10	4.02	2.36	2.13
720.00 LB	724.50	40.57	10.24	5.67	0.45	2.28	3.96
724.50	729.00	190.26	26.28	5.89	2.11	5.84	7.24
729.00	733.50	448.27	41.88	5.22	4.98	9.31	10.70

Errors, Warnings and Notes

Warning - The velocity head has changed by more than 0.5 ft (0.15 m). This may indicate the need for additional cross sections.

Warning - 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.

Flow in subsection defined by left and right stations

Figure 8.13 Example of the Flow Distribution Type of Table

At the bottom of each of the cross section 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.

Cross Section Table Options

Under the **Options** menu of the cross section table window, the user has the following options:

Plans. This option allows the user to select which plan, and therefore output file, they would like to view.

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.

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 Tables

Profile tables are used to show a limited number of hydraulic variables for several cross sections. To display a profile table on the screen, select **Profile Table** from the **View** menu of the main HEC-RAS window. An example profile table is shown in Figure 8.14.

Profile Output Table - Standard Table 1

File Options Std. Tables User Tables Help

HEC-RAS Plan: existing River: Butte Creek Reach: Tributary

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)
Tributary	0.3	150.00	70.10	78.87		78.93	0.000601	1.98	75.70	13.27
Tributary	0.3	250.00	70.10	80.52		80.62	0.000809	2.52	100.72	23.17
Tributary	0.3	500.00	70.10	83.18		83.29	0.000722	2.94	249.73	81.45
Tributary	0.3	750.00	70.10	84.42		84.54	0.000775	3.31	352.24	84.54
Tributary	0.24	150.00	70.00	78.57	72.76	78.63	0.000559	1.93	77.89	13.18
Tributary	0.24	250.00	70.00	80.10	73.75	80.20	0.000824	2.52	99.14	16.53
Tributary	0.24	500.00	70.00	82.77	75.61	82.90	0.000815	3.07	230.07	78.13
Tributary	0.24	750.00	70.00	83.97	77.04	84.12	0.000897	3.49	328.28	83.68
Tributary	0.22	Bridge								
Tributary	0.20	150.00	70.00	78.24	72.75	78.31	0.000647	2.04	73.64	12.87
Tributary	0.20	250.00	70.00	79.84	73.76	79.95	0.000914	2.62	95.48	14.40
Tributary	0.20	500.00	70.00	82.25	75.62	82.42	0.001116	3.46	191.33	70.99
Tributary	0.20	750.00	70.00	83.91	77.05	84.06	0.000927	3.54	323.42	83.53

Total flow in cross section.

Figure 8.14 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 Weir. 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 Weir 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.

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.

To view one of the types of tables, select the desired table type from the **Std. Tables** menu on the profile 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 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 8.15. 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.

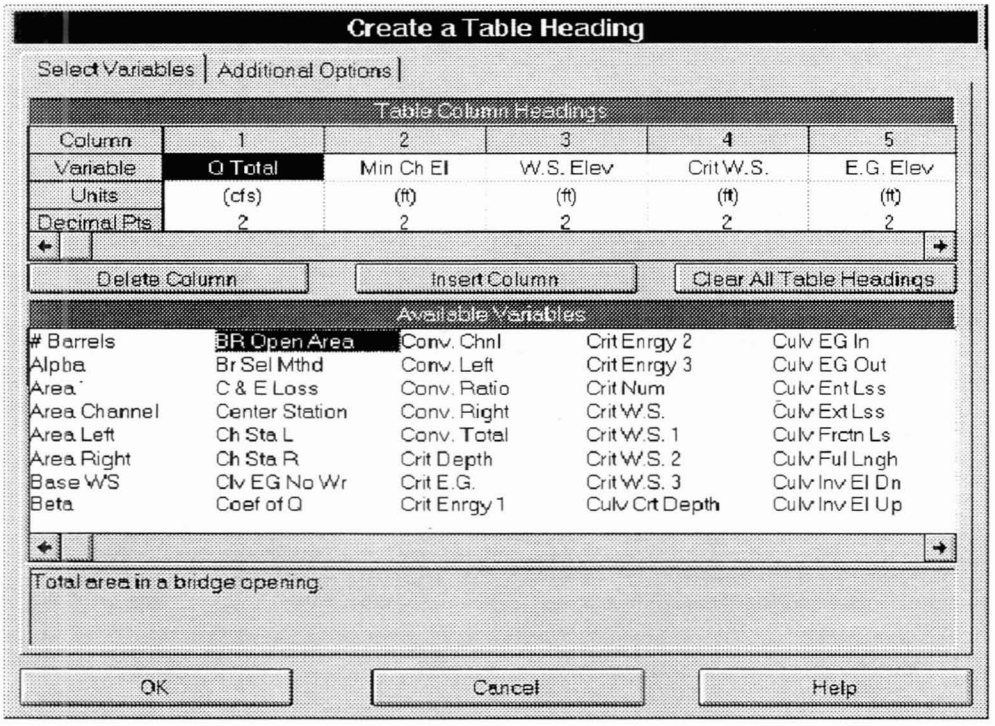


Figure 8.15 User Defined Tables Window

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 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 8.16.

In Figure 8.16, 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.

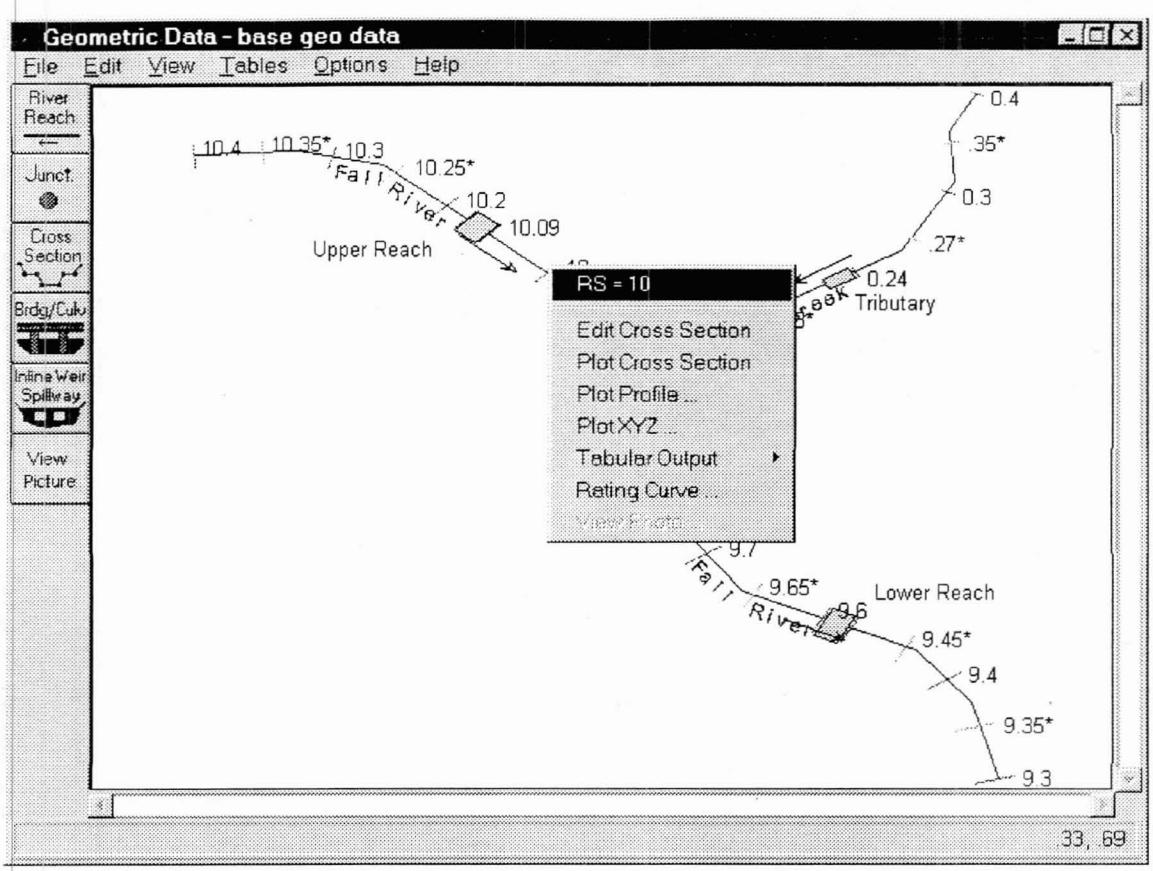


Figure 8.16 Geometric Data Window With Pop up Menu

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 8.17 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.

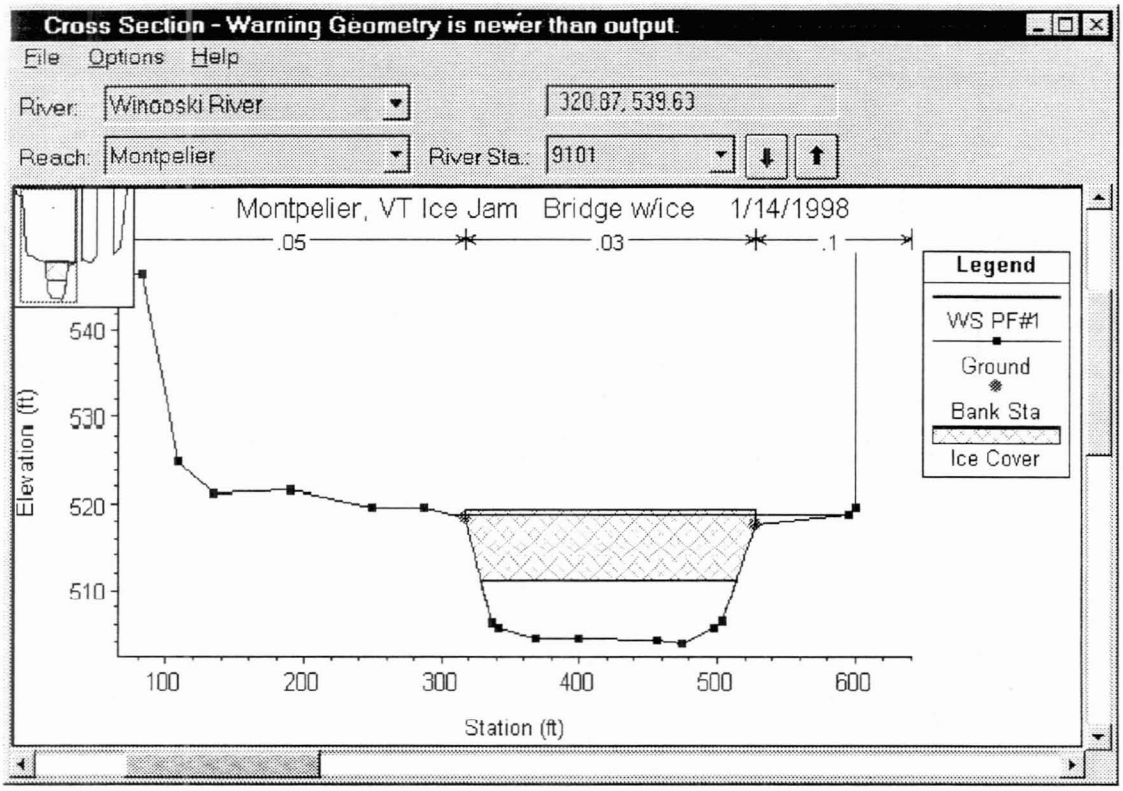


Figure 8.17 Cross section Plot with ice.

Profiles Plot. An example of a profile plot with ice is shown in Figure 8.18. 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.

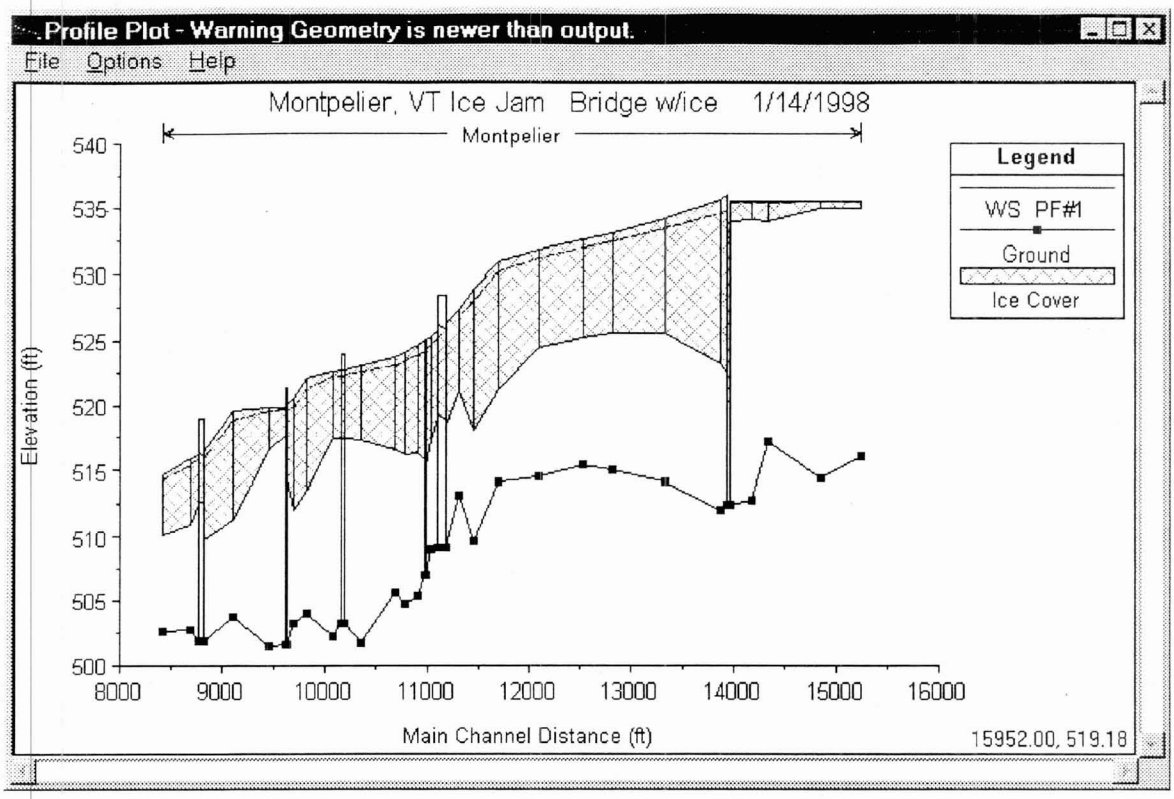


Figure 8.18 Profile plot with ice cover.

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 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 8.19. Tables of ice information can also be created using the **Define Table** option under the **Options** menu of the Profile Table window.

This Page Intentionally Left Blank

CHAPTER 9

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, 1985).

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 9 of the HEC-RAS hydraulics reference manual. This chapter describes how to enter floodplain encroachment data, how to perform the encroachment calculations, and viewing the floodplain encroachment results.

Contents

- General
- Entering Floodplain Encroachment Data
- Performing the Floodplain Encroachment Analysis
- Viewing the Floodplain Encroachment Results

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 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 a detailed discussion on each of these methods, the user is referred to Chapter 9 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 6.1. 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.

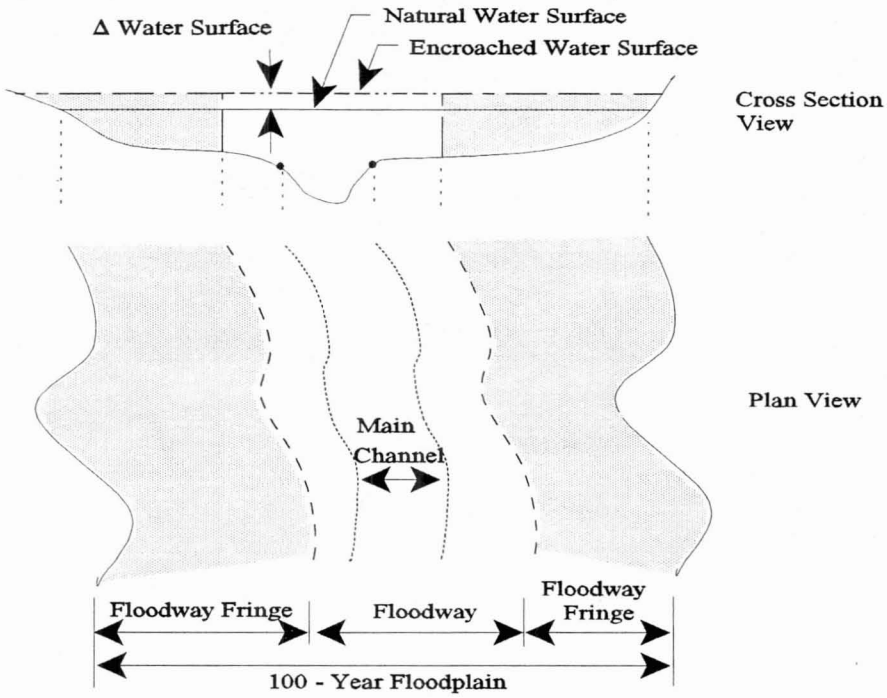


Figure 9.1 Floodway Definition Sketch

Entering Floodplain Encroachment Data

Within HEC-RAS, the data for performing a 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 9.2 (except yours will be blank when you first open it).

As shown in Figure 9.2, 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.

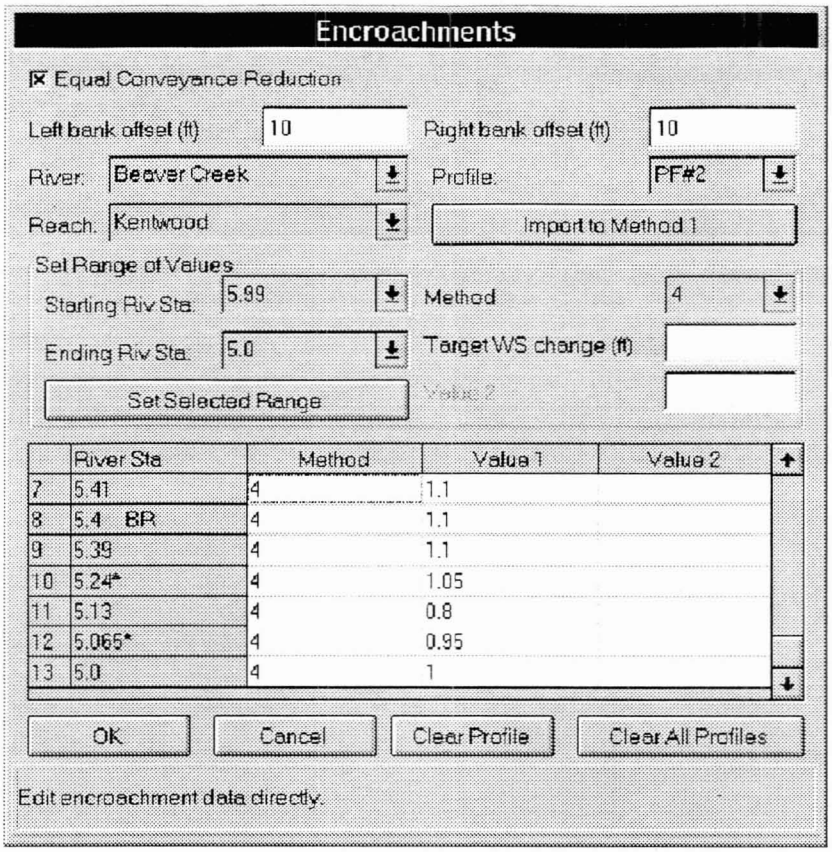


Figure 9.2 Floodplain Encroachment Data Editor

The data for an encroachment analysis should be entered in the following manner:

Global Information. Global information are 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 topwidth
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 9.3.

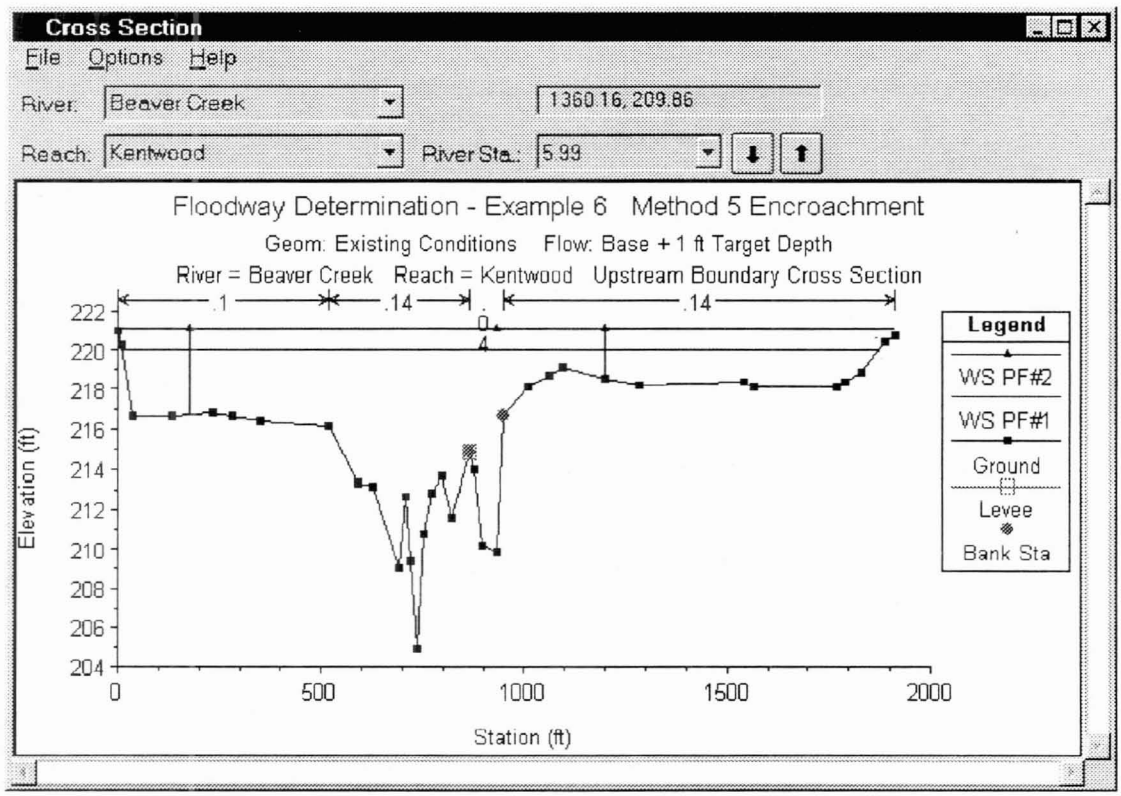


Figure 9.3 Example Cross Section Plot With Encroachments

As shown in Figure 9.3, 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 9.4. 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.

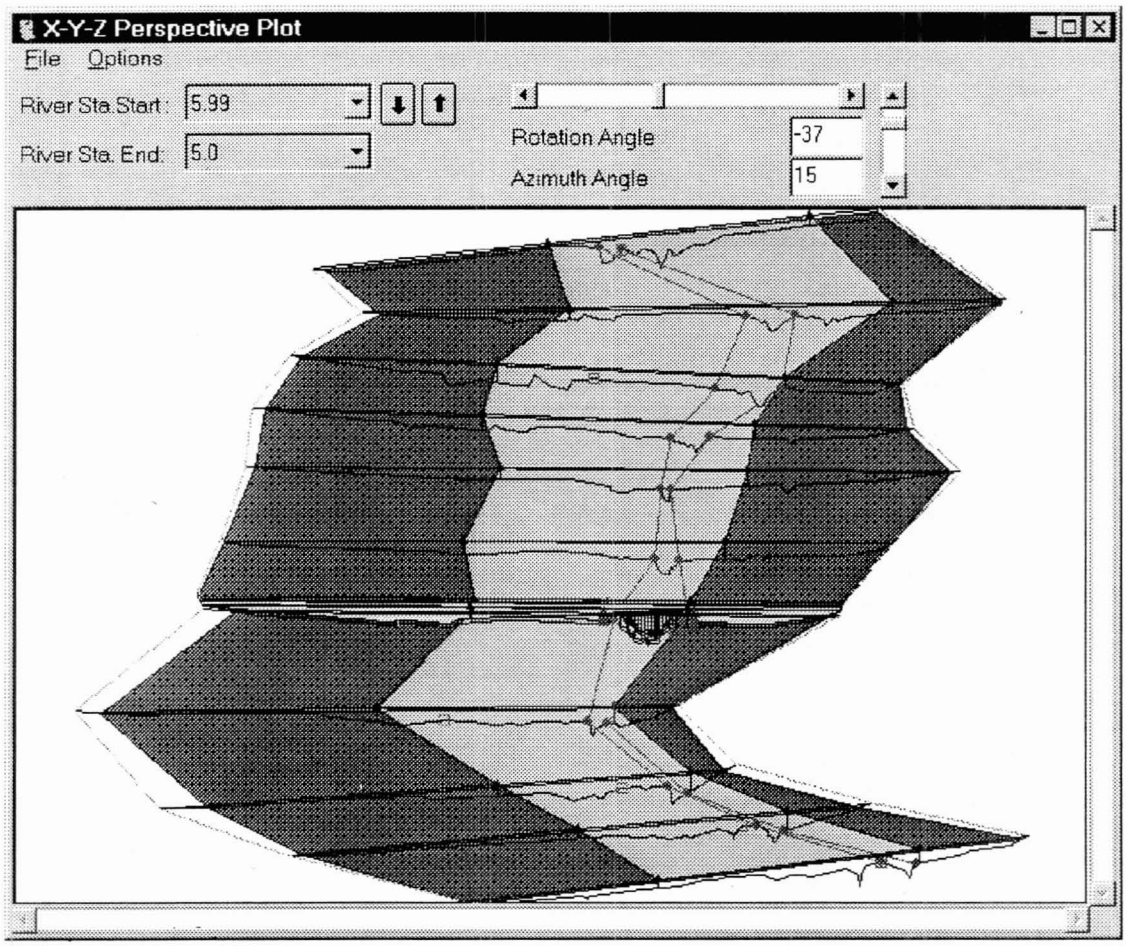


Figure 9.4 Example X-Y-Z Perspective Plot with Base and Encroached Profiles

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 9.5. 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 User Tables Help

HEC-RAS Plan: M4 - Trial 3 River: Beaver Creek Reach: Kentwood

Reach	River Sta	W.S. Elev	Prof Delta WS	E.G. Elev	Top Wdth Act	Q Left	Q Channel	Q Right	Enc Sta L	Ch Sta L
		(ft)	(ft)	(ft)	(ft)	(cfs)	(cfs)	(cfs)	(ft)	(ft)
Kentwood	5.4 BR U	217.42		217.66	1846.93	826.94	10817.04	2312.15		450.00
Kentwood	5.4 BR U	217.54	0.12	218.01	617.70	32.04	12557.45	1397.33	440.00	450.00
Kentwood	5.4 BR D	217.42		217.66	1824.00	835.94	10864.89	2255.30		450.00
Kentwood	5.4 BR D	217.54	0.12	218.01	617.70	32.72	12577.22	1376.87	440.00	450.00
Kentwood	5.39	215.62		216.04	1702.86	1073.64	10252.97	2673.38		450.00
Kentwood	5.39	216.61	1.00	217.23	617.70	30.52	12569.51	1399.97	440.00	450.00
Kentwood	5.24*	214.64		214.77	1633.32	2335.79	2506.46	9157.75		200.30
Kentwood	5.24*	215.64	1.00	215.83	680.87	322.52	3058.80	10618.67	182.33	200.30
Kentwood	5.13	213.33		213.76	1429.70	1102.71	4951.68	7945.61		155.00
Kentwood	5.13	214.33	1.00	214.84	595.57	203.94	5543.43	8252.63	145.00	155.00
Kentwood	5.065*	212.54		212.88	1781.77	1624.37	5359.40	7016.23		274.50
Kentwood	5.065*	213.55	1.00	214.01	674.15	165.81	6420.46	7413.72	264.50	274.50
Kentwood	5.0	211.80		212.05	1925.36	2217.73	5187.01	6595.26		394.00
Kentwood	5.0	212.80	1.00	213.18	912.57	127.69	6544.90	7327.41	384.00	394.00

Calculated water surface from energy equation.

Figure 9.5 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 9.6.

Profile Output Table - Encroachment 2

File Options Std. Tables User Tables Help

HEC-RAS Plan: M4 - Trial 3 River: Beaver Creek Reach: Kentwood

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
		(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	(ft)
Kentwood	5.99		1862.53				907.00		
Kentwood	5.99	1.00	885.42	8.55	233.00	674.00	907.00	211.42	1118.42
Kentwood	5.875*		1797.30				678.25		
Kentwood	5.875*	0.99	928.58	12.48	327.07	351.18	678.25	577.40	1255.65
Kentwood	5.76		1764.76				449.50		
Kentwood	5.76	0.86	847.98	14.08	341.00	108.50	449.50	739.48	1188.98
Kentwood	5.685*		1872.46				647.25		
Kentwood	5.685*	0.72	789.38	18.42	464.82	182.43	647.25	606.95	1254.20
Kentwood	5.61		1987.63				845.00		
Kentwood	5.61	0.63	707.61	21.55	609.60	235.40	845.00	472.22	1317.22
Kentwood	5.49*		1910.01				647.00		
Kentwood	5.49*	0.44	752.77	20.62	477.33	169.67	647.00	583.10	1230.10
Kentwood	5.41		1846.93				548.50		
Kentwood	5.41	0.12	617.70		440.00	108.50	548.50	509.20	1057.70

Difference in WS between current profile and WS for first profile

Figure 9.6 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 9.7

Profile Output Table - Encroachment 3

File Options Std. Tables User Tables Help

HEC-RAS Plan: M4 - Trial 3 River: Beaver Creek Reach: Kentwood

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	1862.53	6604.01	2.12	220.00	220.00	
Kentwood	5.99	885.42	5563.61	2.52	221.00	220.00	1.00
Kentwood	5.875*	1797.30	7058.81	1.98	218.99	218.99	
Kentwood	5.875*	928.58	5276.33	2.65	219.97	218.99	0.99
Kentwood	5.76	1764.76	9080.01	1.54	218.45	218.45	
Kentwood	5.76	847.98	5912.37	2.37	219.32	218.45	0.86
Kentwood	5.685*	1872.46	9233.37	1.52	218.22	218.22	
Kentwood	5.685*	789.38	5529.23	2.53	218.94	218.22	0.72
Kentwood	5.61	1987.63	9481.46	1.48	218.09	218.09	
Kentwood	5.61	707.61	5351.25	2.62	218.71	218.09	0.63
Kentwood	5.49*	1910.01	9428.42	1.48	217.90	217.90	
Kentwood	5.49*	752.77	5178.39	2.70	218.33	217.90	0.44
Kentwood	5.41	1846.93	8960.55	1.56	217.42	217.42	
Kentwood	5.41	617.70	4042.22	3.46	217.54	217.42	0.12
Kentwood	5.4 BR U	1846.93	2508.57	5.58	217.42	217.42	
Kentwood	5.4 BR U	617.70	1978.13	7.08	217.54	217.42	0.12
Kentwood	5.4 BR D	1824.00	2497.48	5.61	217.42	217.42	
Kentwood	5.4 BR D	617.70	1978.13	7.08	217.54	217.42	0.12

Top width of the wetted cross section, not including ineffective flow.

Figure 9.7 Example of the Encroachment 3 Standard Table

This Page Intentionally Left Blank

CHAPTER 10

Troubleshooting With HEC-RAS

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.

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 computations. When the user presses the compute button, on the steady flow analysis window, the entire data set is processed through several data checks before the computations are processed. 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 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 10.1.

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.

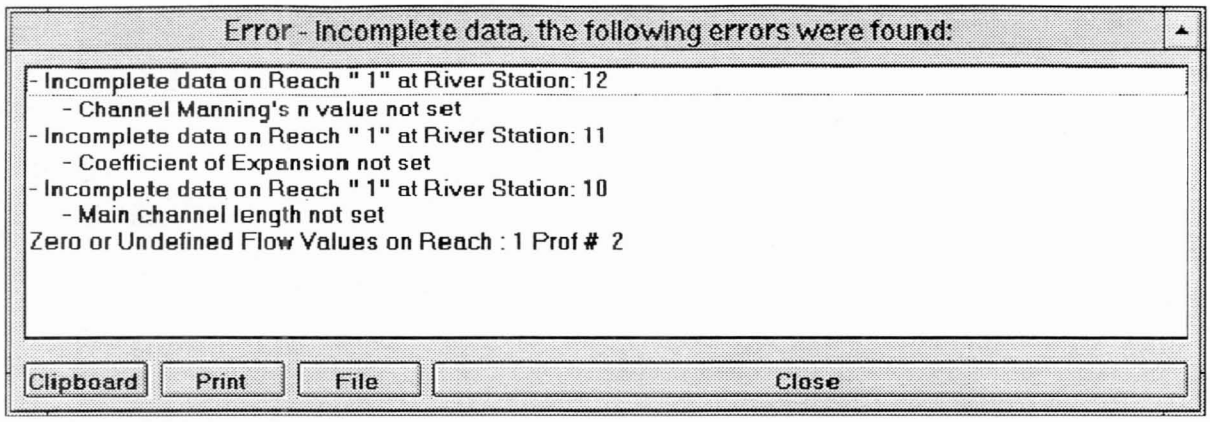


Figure 10.1 Data Completeness Checking Window

Errors, Warnings, and Notes

The HEC-RAS software has a system of Errors, Warnings, and Notes that are passed from the steady flow computation program to the user interface. During the computations, the steady flow computation program 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 10.2.

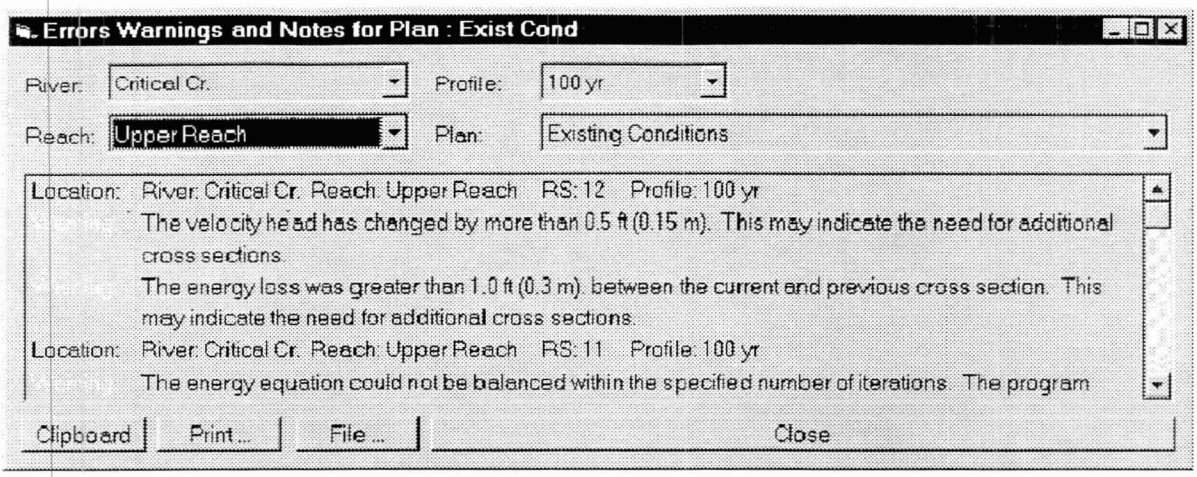


Figure 10.2 Summary of Errors, Warnings, and Notes Window

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 10.3.

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:

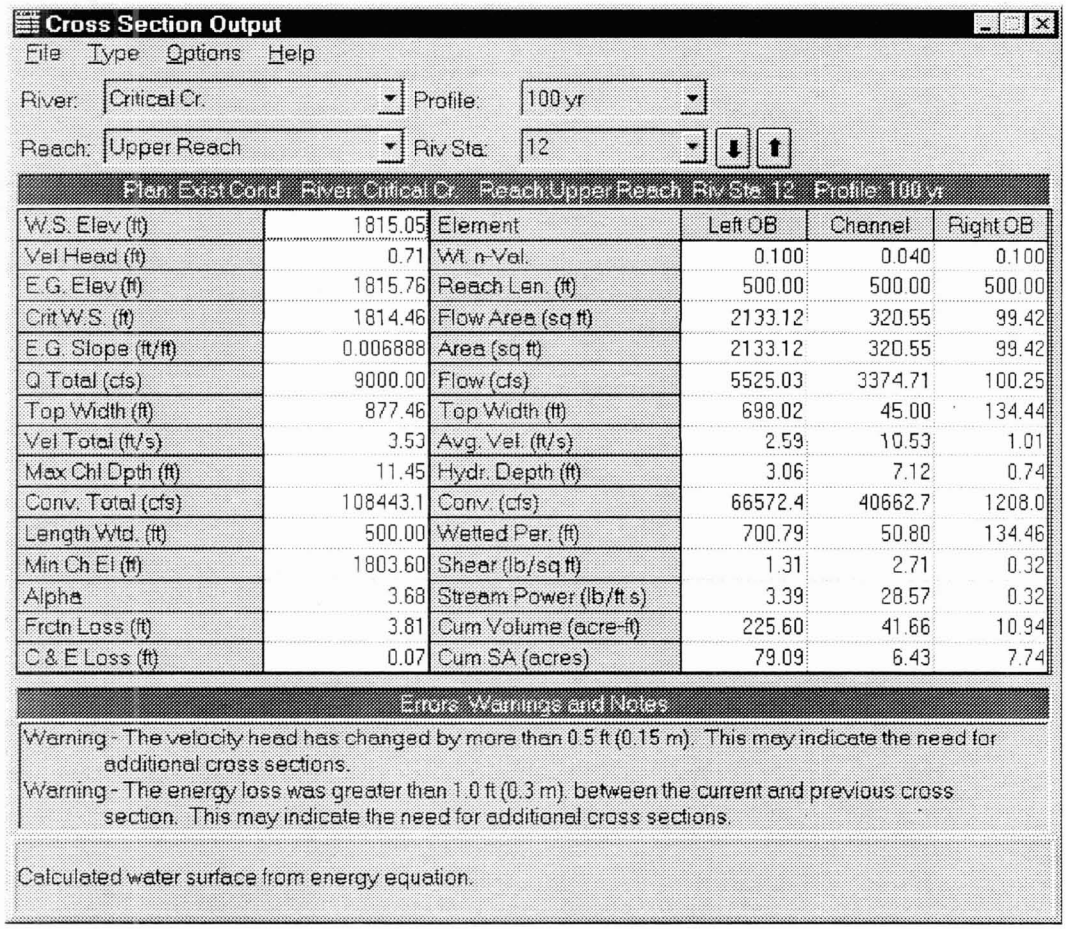
Cross sections spaced to far apart. This can cause several warning messages to be set.

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 Riv Sta: 12

Plan: Exist Cond River: Critical Cr. Reach: Upper Reach Riv Sta: 12 Profile: 100 yr

W.S. Elev (ft)	1815.05	Element	Left OB	Channel	Right OB
Vel Head (ft)	0.71	Wt. n-Val.	0.100	0.040	0.100
E.G. Elev (ft)	1815.76	Reach Len. (ft)	500.00	500.00	500.00
Crit W.S. (ft)	1814.46	Flow Area (sq ft)	2133.12	320.55	99.42
E.G. Slope (ft/ft)	0.006888	Area (sq ft)	2133.12	320.55	99.42
Q Total (cfs)	9000.00	Flow (cfs)	5525.03	3374.71	100.25
Top Width (ft)	877.46	Top Width (ft)	698.02	45.00	134.44
Vel Total (ft/s)	3.53	Avg. Vel. (ft/s)	2.59	10.53	1.01
Max Chl Dpth (ft)	11.45	Hydr. Depth (ft)	3.06	7.12	0.74
Conv. Total (cfs)	108443.1	Conv. (cfs)	66572.4	40662.7	1208.0
Length Wtd. (ft)	500.00	Wetted Per. (ft)	700.79	50.80	134.46
Min Ch El (ft)	1803.60	Shear (lb/sq ft)	1.31	2.71	0.32
Alpha	3.68	Stream Power (lb/ft s)	3.39	28.57	0.32
Frctn Loss (ft)	3.81	Cum Volume (acre-ft)	225.60	41.66	10.94
C & E Loss (ft)	0.07	Cum SA (acres)	79.09	6.43	7.74

Errors, Warnings and Notes

Warning - The velocity head has changed by more than 0.5 ft (0.15 m). This may indicate the need for additional cross sections.

Warning - 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.

Calculated water surface from energy equation.

Figure 10.3 Cross Section Table with Errors, Warnings, and Notes

Log Output

Setting 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 10.4. 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.

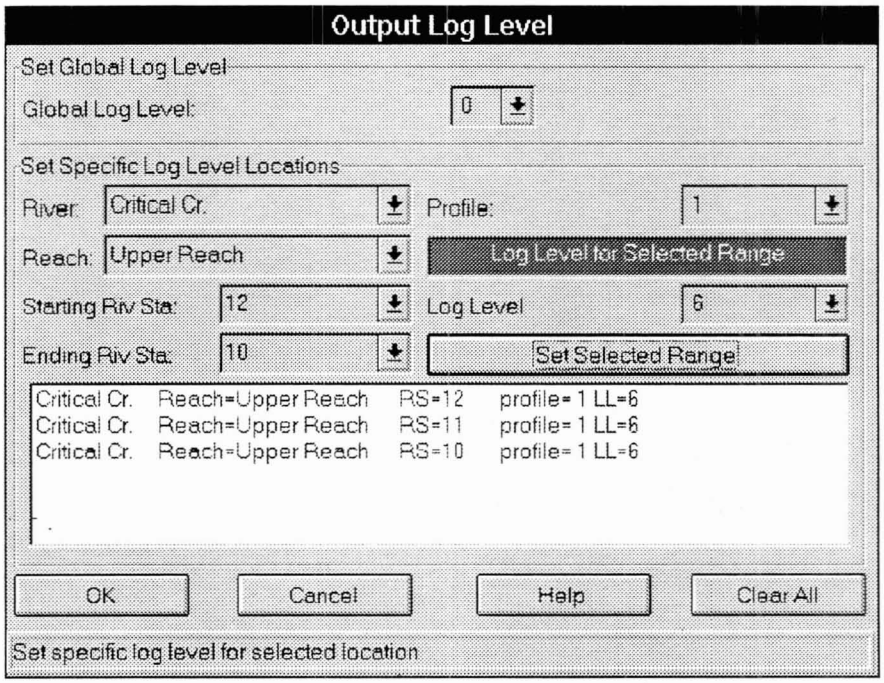


Figure 10.4 Log File Output Level window

Viewing The 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.

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 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 cross section specific tables to get detailed results at a single location.

The Occurrence of Critical Depth

During the 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. 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.

3. 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.

4. 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.

Computational Program Does Not Run To Completion

While running the computational part of the software, when the program is finished you should get the message "PROGRAM TERMINATED NORMALLY." 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, as shown in Figure 10.5.

This is a Fortran error message that is put out by 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 almost always a data input problem. There is a "small" 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."

 Chapter 10 Troubleshooting With HEC-RAS

```
[Inactive Steady Flow Analysis]

PROFILE     1

                                EG        WSEL    CRITICAL

       XSEC  93.2      =   661.54      661.50      654.11
       XSEC  93.4      =   661.84      661.73      653.78
  BRIDGE
D:\nexgen\Fp4\snet\Areawp.for(238) : run-time error M6104: MATH
- floating-point error: overflow
```

Figure 10.5 Steady Flow Analysis Computation Window

The first step in finding the problem is to realize where the error is occurring. From the information in the computation window, shown in Figure 10.5, the program was on the first profile and had finished computing the water surface and energy for cross section 93.4. This means that the error occurred at the next computational point upstream of section 93.4. Go to the Geometric Data editor and review the input data closely at the problem location.

Computational errors, such as the one shown in Figure 10.5, 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.

This Page Intentionally Left Blank

CHAPTER 11

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, 1995). 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 11-1.

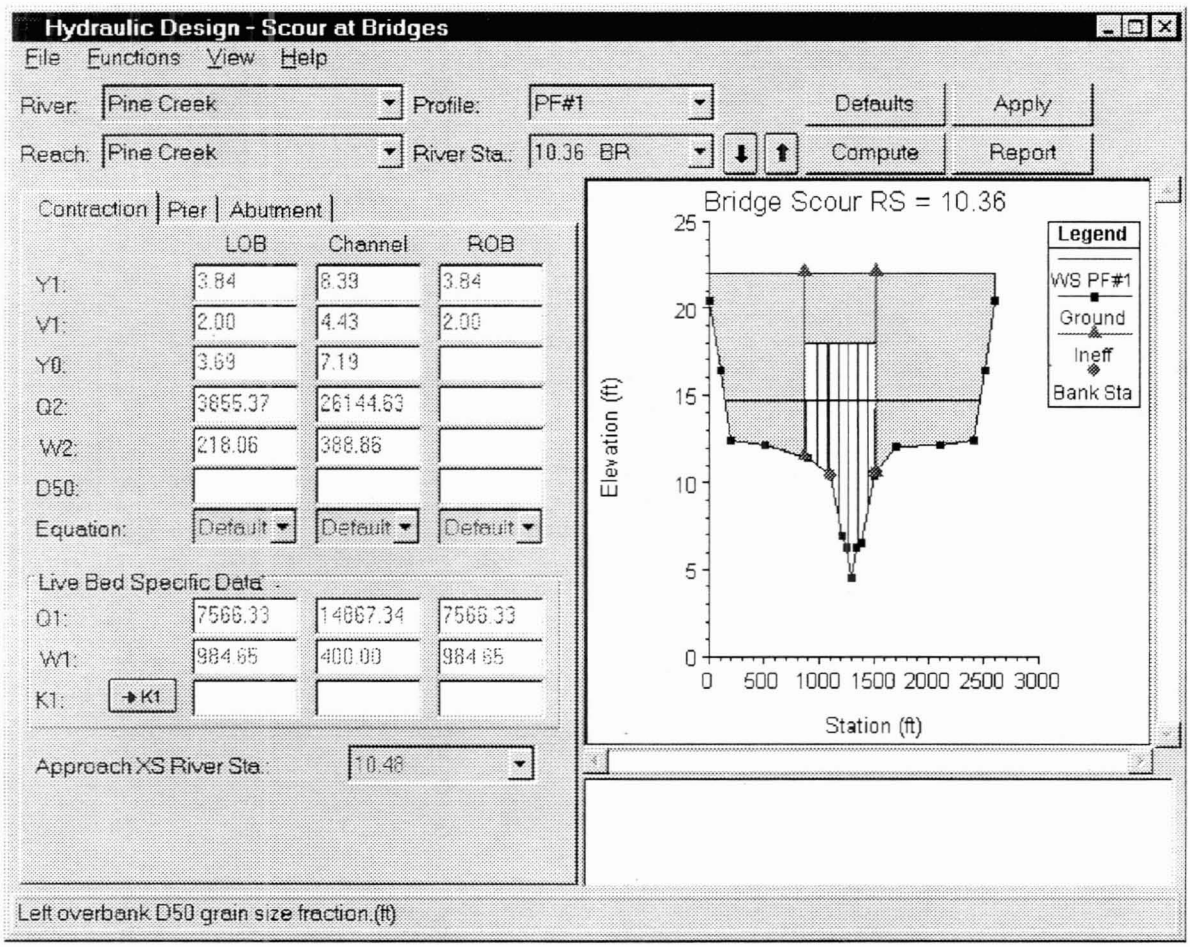


Figure 11-1. Hydraulic Design Window For Scour at Bridges

As shown in Figure 11-1, 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 11-2 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.

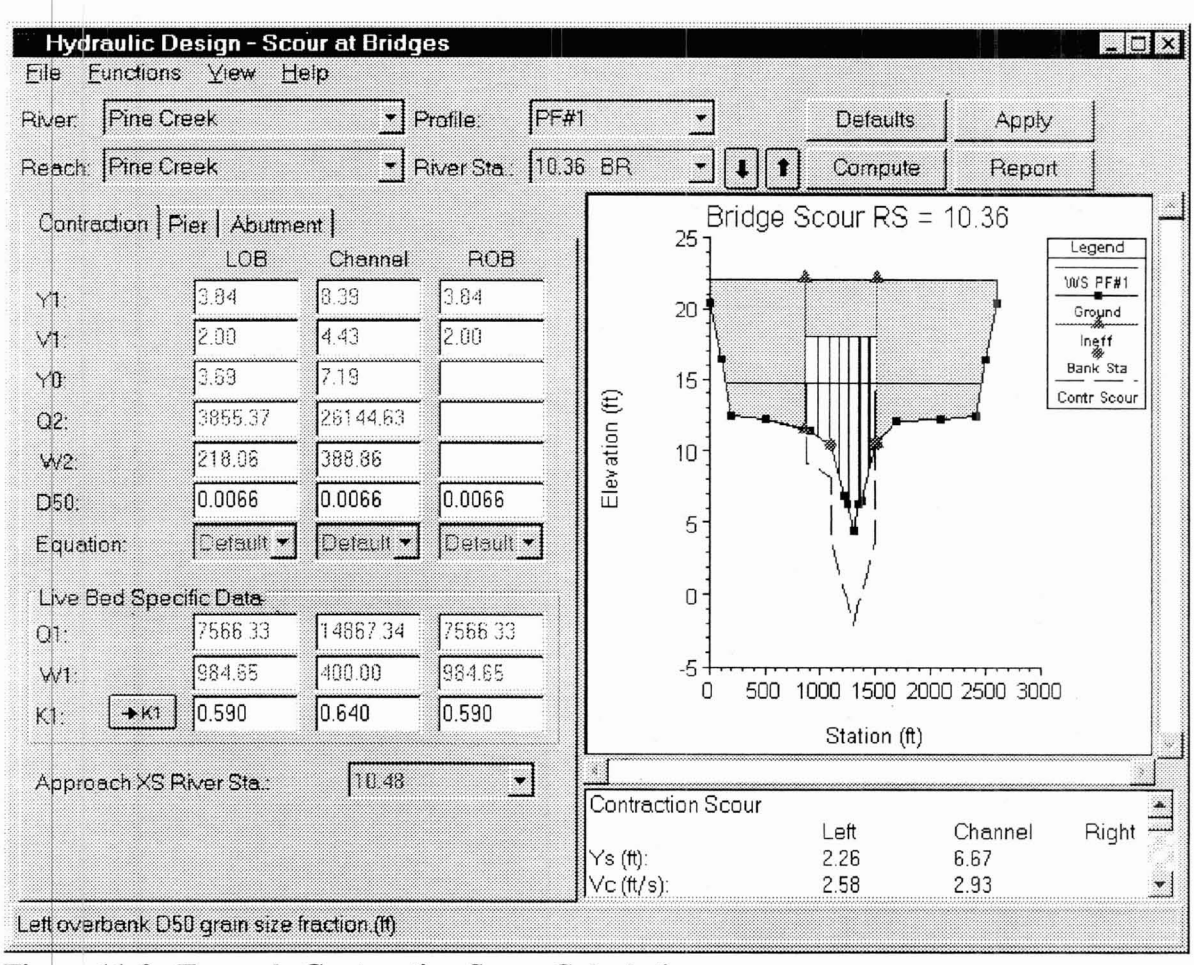


Figure 11-2. Example Contraction Scour Calculation

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 values 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 11-3 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.

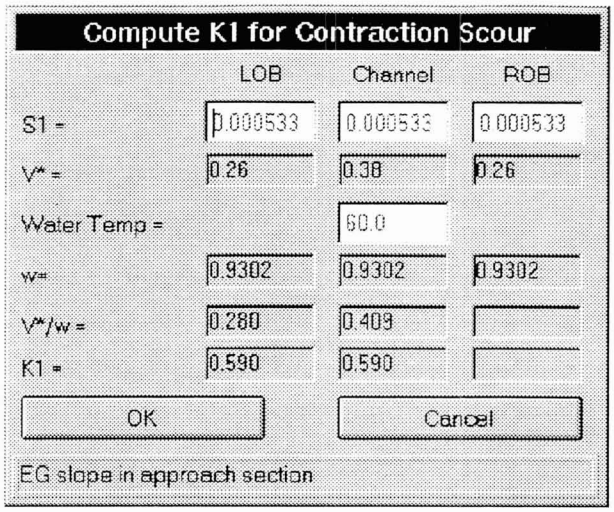


Figure 11-3. Computation of the K1 Factor.

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 11-2, 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 11-4, 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 D90 size fraction for the bed material. All other values are automatically obtained from the HEC-RAS output file.

As shown in Figure 11-4, 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.

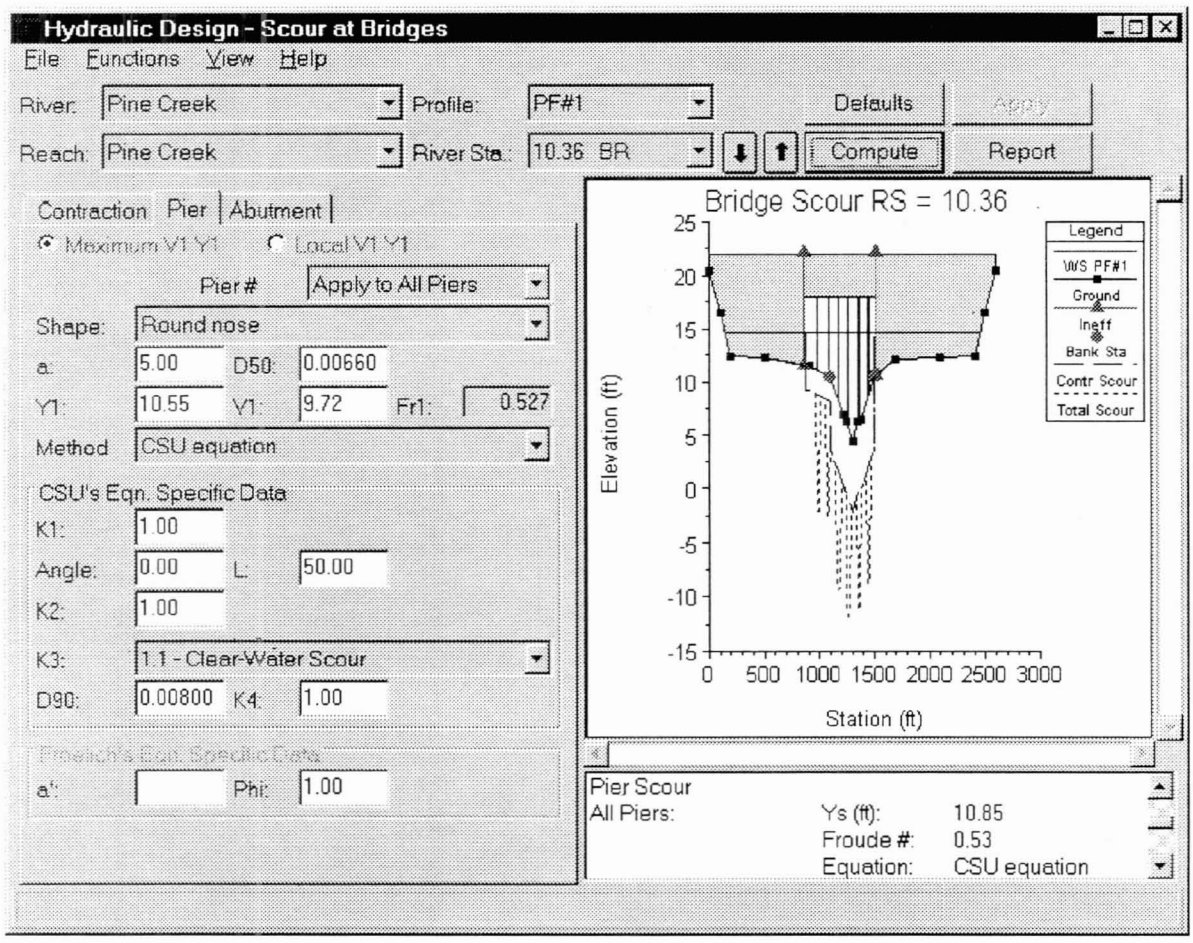


Figure 11-4. Example Pier Scour Computation.

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.

D90: The median size of the bed material of which 90 percent is finer. The D90 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 0.2 feet (0.06 m). This factor is automatically calculated by the program, and is a function of D50; D90; 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 11-4 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 11-4). Also shown in Figure 11-4 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 11-5.

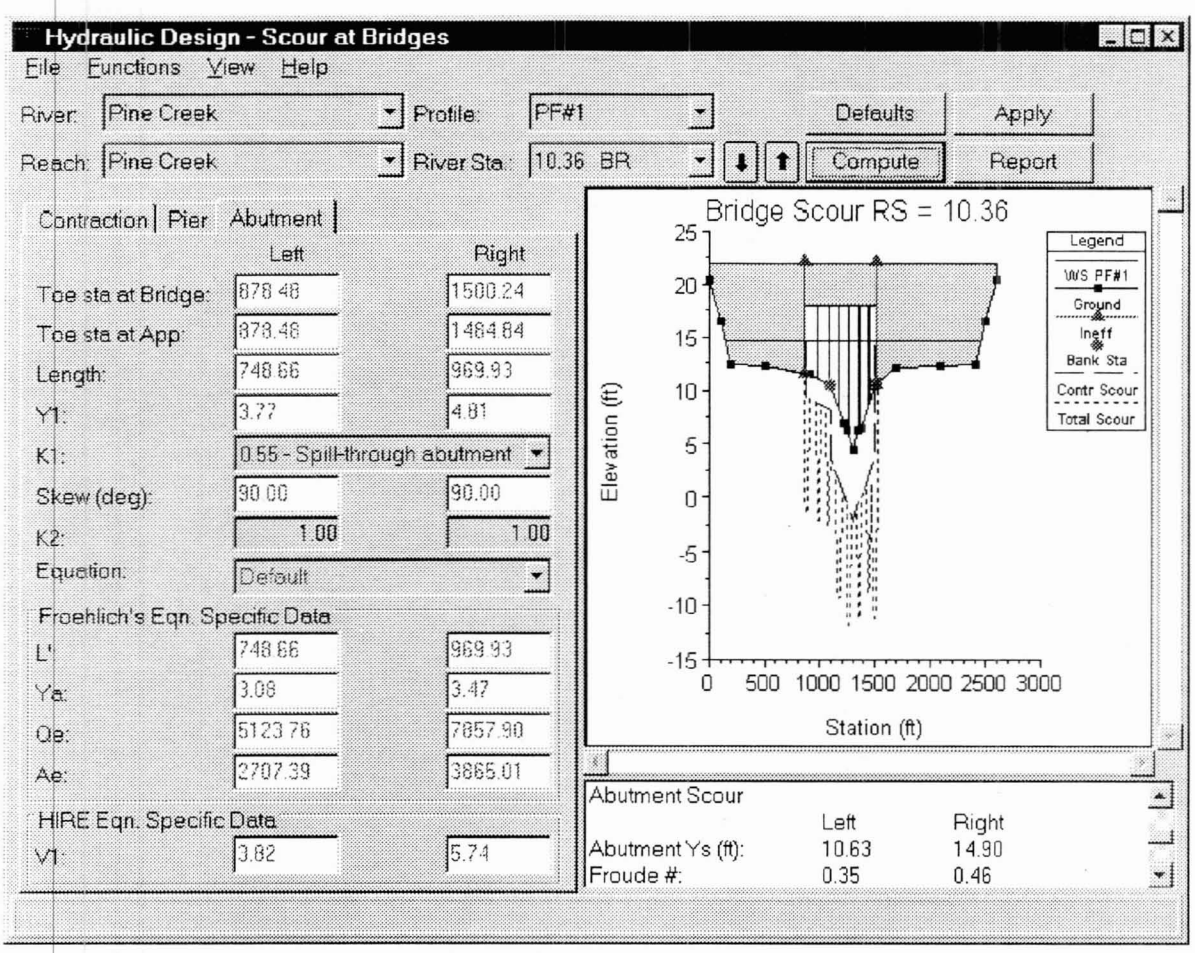


Figure 11-5. Example Abutment Scour Computations

As shown in Figure 11-5, 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 minus the station of the left extent of the water surface in the cross section just upstream of the bridge (including ineffective flow area). The right embankment length is computed as the stationing of the right extent of the water surface minus the stationing of the toe of the right abutment, at the cross section just upstream of the bridge. 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. 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 directly.

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 11-5, the program selected the HIRE equation and computed 10.63 feet (3.24 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 11.1 shows a summary of the computed results, including the total scour.

Table 11.1
Summary of Scour Computations

Contraction Scour

	Left O.B.	Main Channel	Right O.B.
Ys =	2.26 ft (0.69 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 =	10.63 ft (3.24 m)	14.90 ft (4.54 m)
Eqn =	HIRE equation	HIRE equation

Total Scour

Left Abutment	=	12.89 ft (3.93 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 11-6 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.

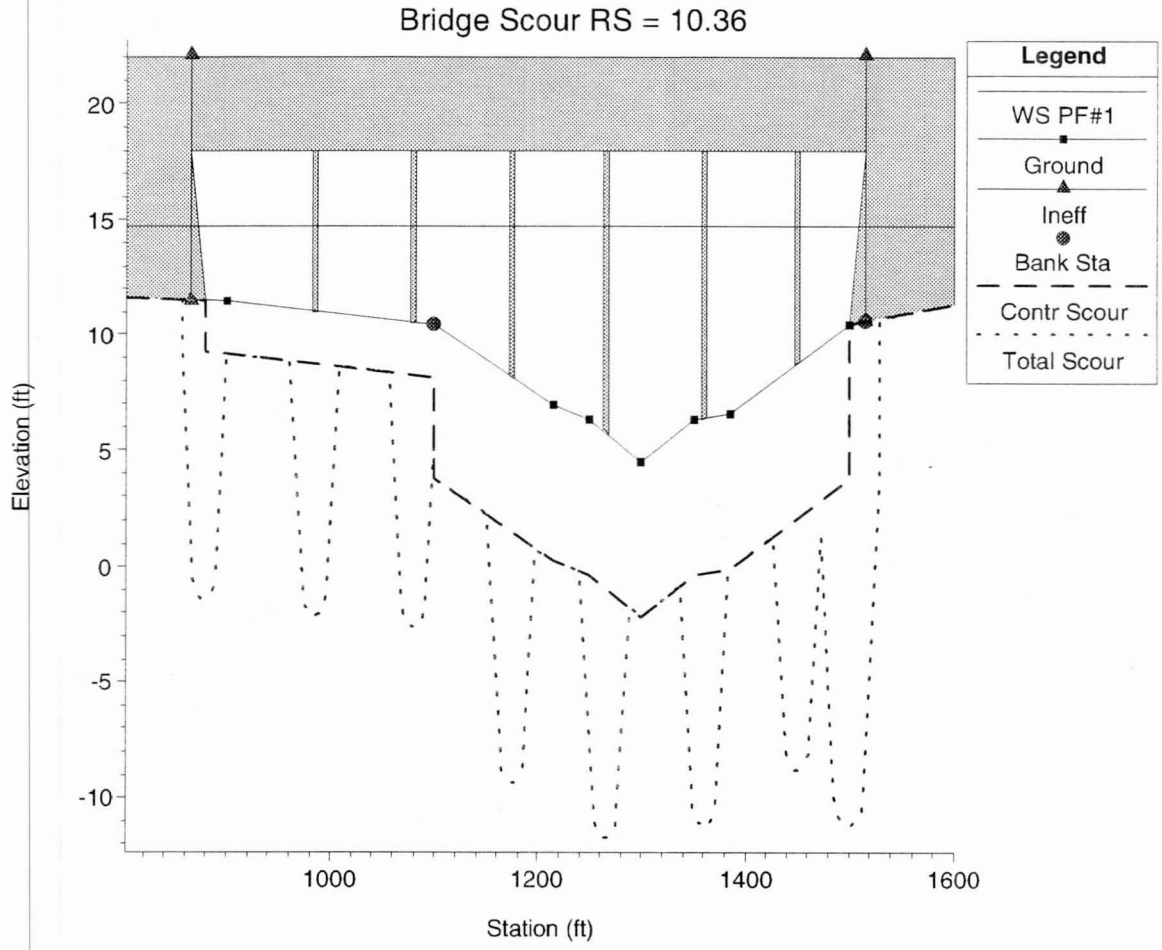


Figure 11-6. Total Scour Depicted in Graphical Form

CHAPTER 12

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 **Options** menu of the Geometric Data window. When this option is selected, a Channel Modification window will appear as shown in Figure 12.1 (except yours will not have any data in it the first time you bring it up).

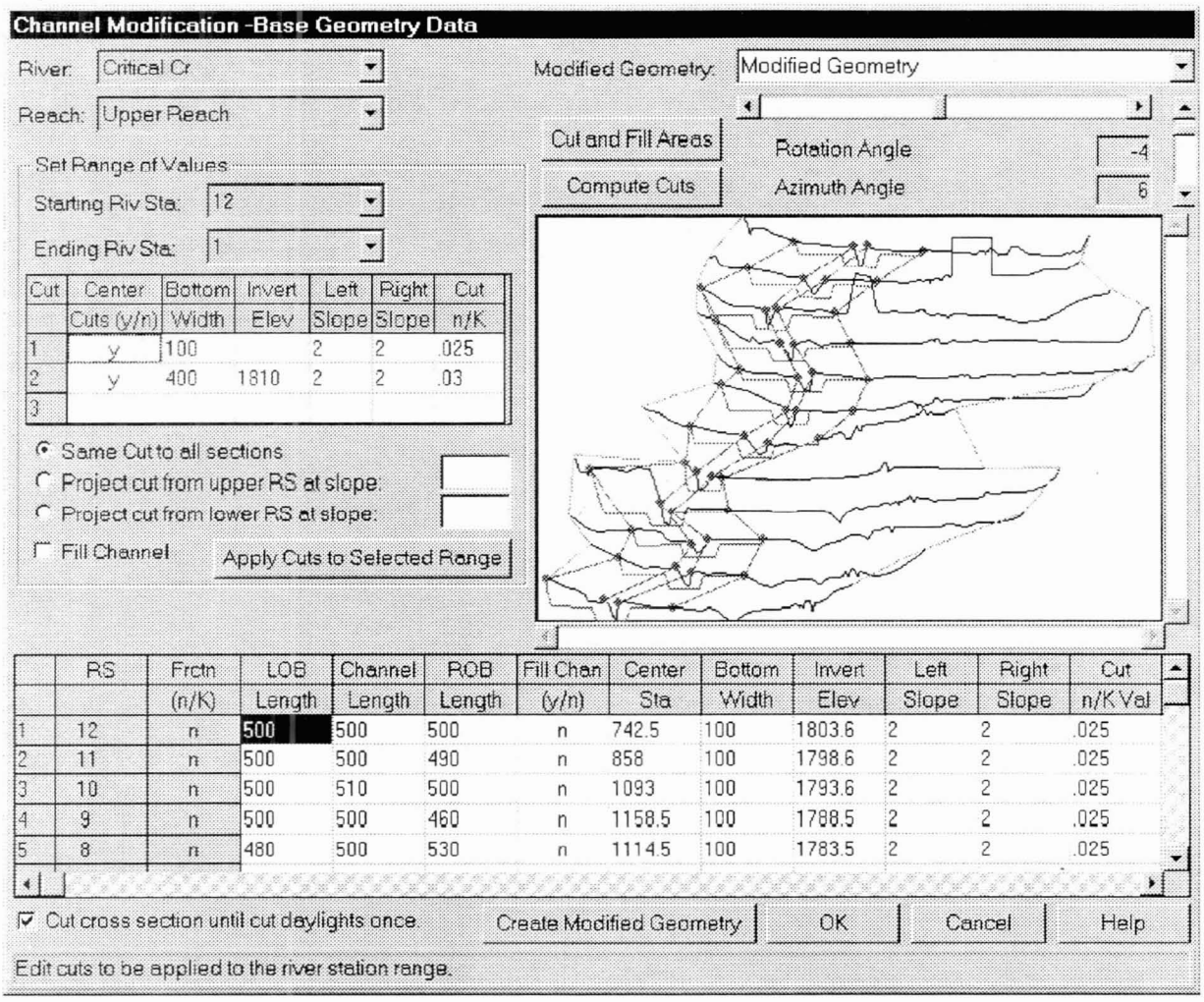


Figure 12.1 Channel Modification Data Editor

As shown in Figure 12.1, 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 12.2.

Channel Modification - Cut and Fill Data

River:	Critical Cr.	Reach:	Upper						
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 12.2 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. This 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 12.3. Figure 12.3 shows the geometry of the modified channel condition, along with the computed water surface elevations from both the existing and modified plans. To display the results from more than one plan on a graphic, the user can select **Plan** from the **Options** menu on any of the graphics.

In addition to graphical output, the user can review the computed results from both plans in a tabular form. Figure 12.4 shows the computed results for both plans in Standard Table 1 of the Profile Output table.

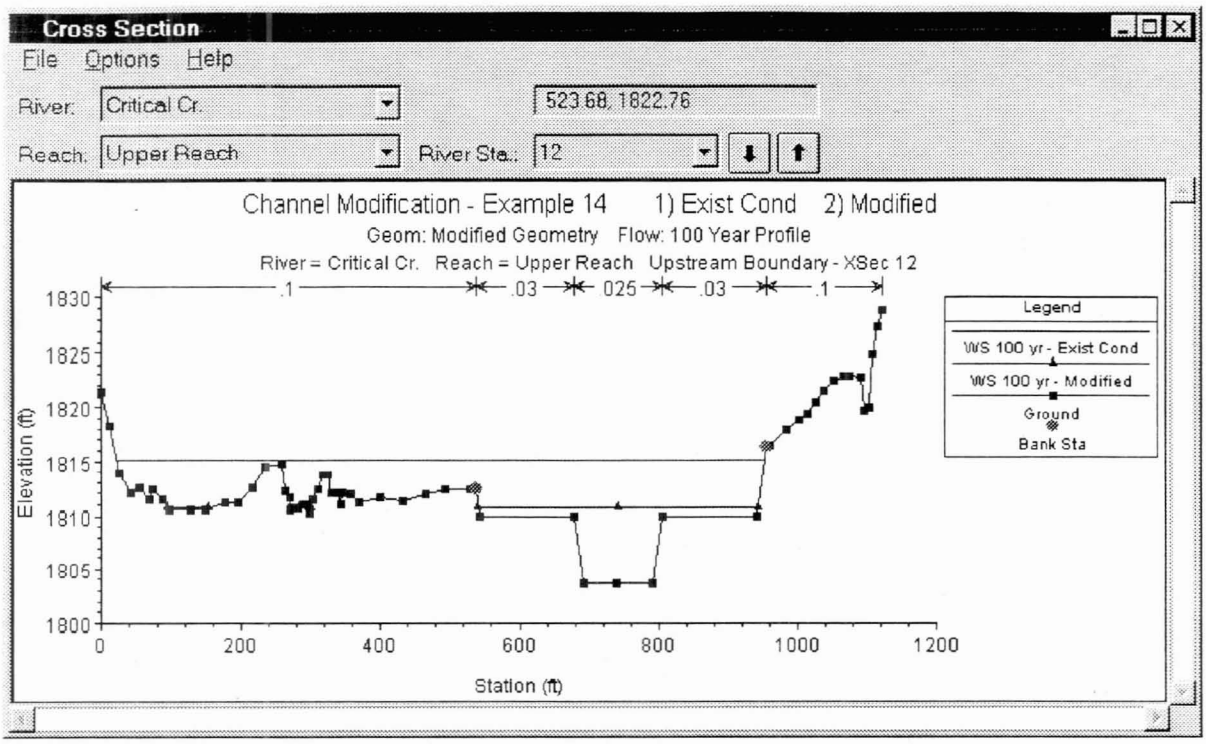
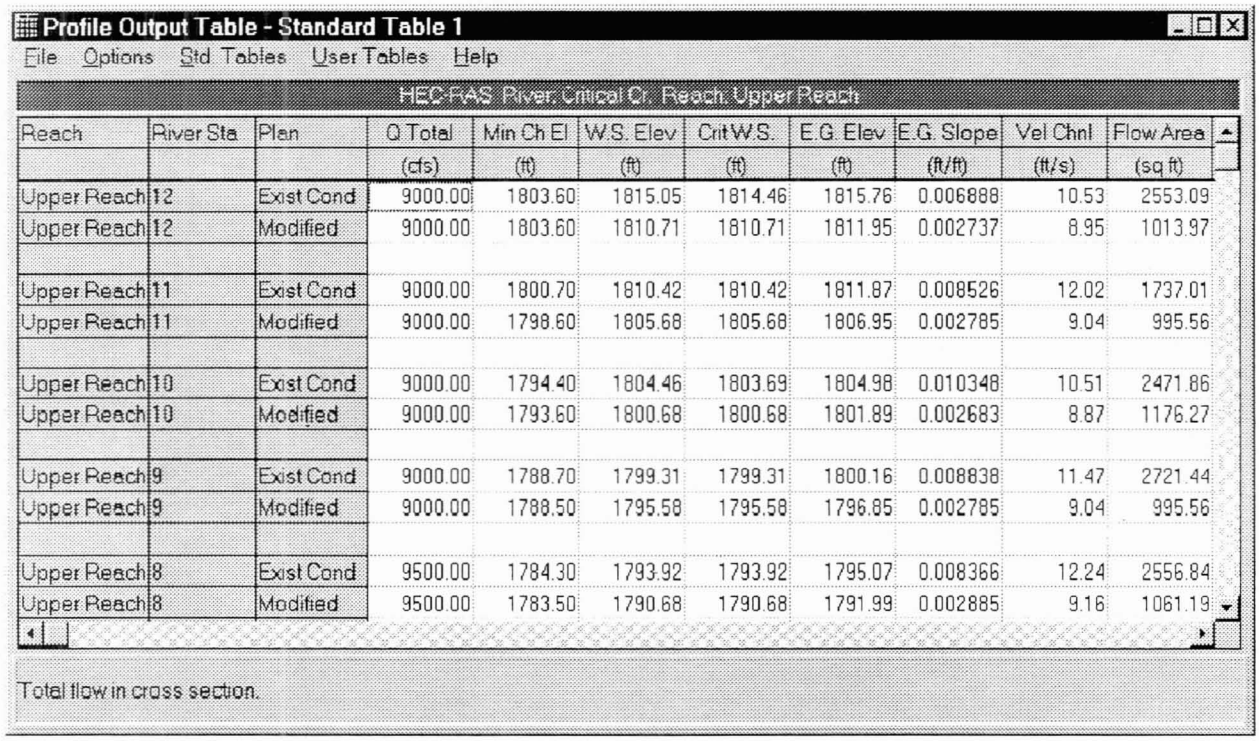


Figure 12.3 Modified Cross Section With Existing and Modified Condition Water Surface



Profile Output Table - Standard Table 1

File Options Std Tables User Tables Help

HEC-RAS River: Critical Cr. Reach: Upper Reach

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	1815.05	1814.46	1815.76	0.006888	10.53	2553.09
Upper Reach	12	Modified	9000.00	1803.60	1810.71	1810.71	1811.95	0.002737	8.95	1013.97
Upper Reach	11	Exist Cond	9000.00	1800.70	1810.42	1810.42	1811.87	0.008526	12.02	1737.01
Upper Reach	11	Modified	9000.00	1798.60	1805.68	1805.68	1806.95	0.002785	9.04	995.56
Upper Reach	10	Exist Cond	9000.00	1794.40	1804.46	1803.69	1804.98	0.010348	10.51	2471.86
Upper Reach	10	Modified	9000.00	1793.60	1800.68	1800.68	1801.89	0.002683	8.87	1176.27
Upper Reach	9	Exist Cond	9000.00	1788.70	1799.31	1799.31	1800.16	0.008838	11.47	2721.44
Upper Reach	9	Modified	9000.00	1788.50	1795.58	1795.58	1796.85	0.002785	9.04	995.56
Upper Reach	8	Exist Cond	9500.00	1784.30	1793.92	1793.92	1795.07	0.008366	12.24	2556.84
Upper Reach	8	Modified	9500.00	1783.50	1790.68	1790.68	1791.99	0.002885	9.16	1061.19

Total flow in cross section.

Figure 12.4 Standard Table 1 With Existing and Modified Conditions

This Page Intentionally Left Blank

CHAPTER 13

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 floodplain 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 a set of macros for Arc Info (using Arc Macro Language, AML) that will allow a user to write the geometric data in the required format, as well as read the HEC-RAS results and perform the floodplain mapping. These macros are not part of the HEC-RAS package. The macros and a user's manual will be provided on an as needed basis. Also, the Intergraph Corporation will be adding the capability to exchange data with HEC-RAS in their Inroads Software package, as well as a new program called Storm Works (Intergraph, 1997)

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.

At this time, Manning's n values, contraction and expansion coefficients, optional cross section properties (ineffective flow areas, levees, 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 the **Import GIS Data** option 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 is working on a set of macros written in AML, to be used with Arc Info. Likewise, the Intergraph Corporation is adding this capability to their Inroads software package, as well as a program called Storm Works. You have the option of obtaining the AML macros from HEC (for use in Arc Info); using Inroads or Storm Works from 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 13.1) in which the user can select the file that contains the geometry data from the GIS.

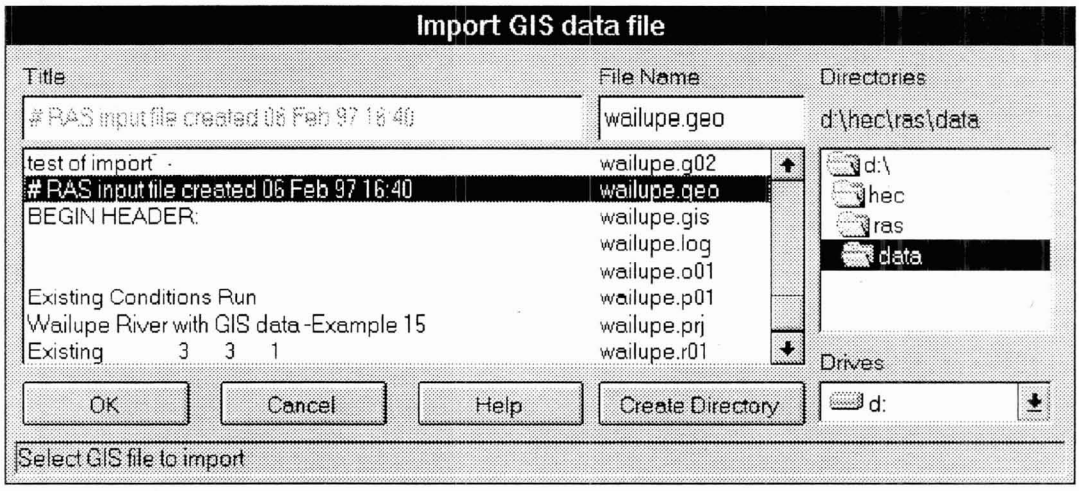


Figure 13.1 Window for Selecting GIS Data File To Import

4. Once the user selects the file containing the GIS data, and then presses the **OK** button, the data will be imported and a schematic of the river system will show up in the Geometric Data window (Figure 13.2). 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.

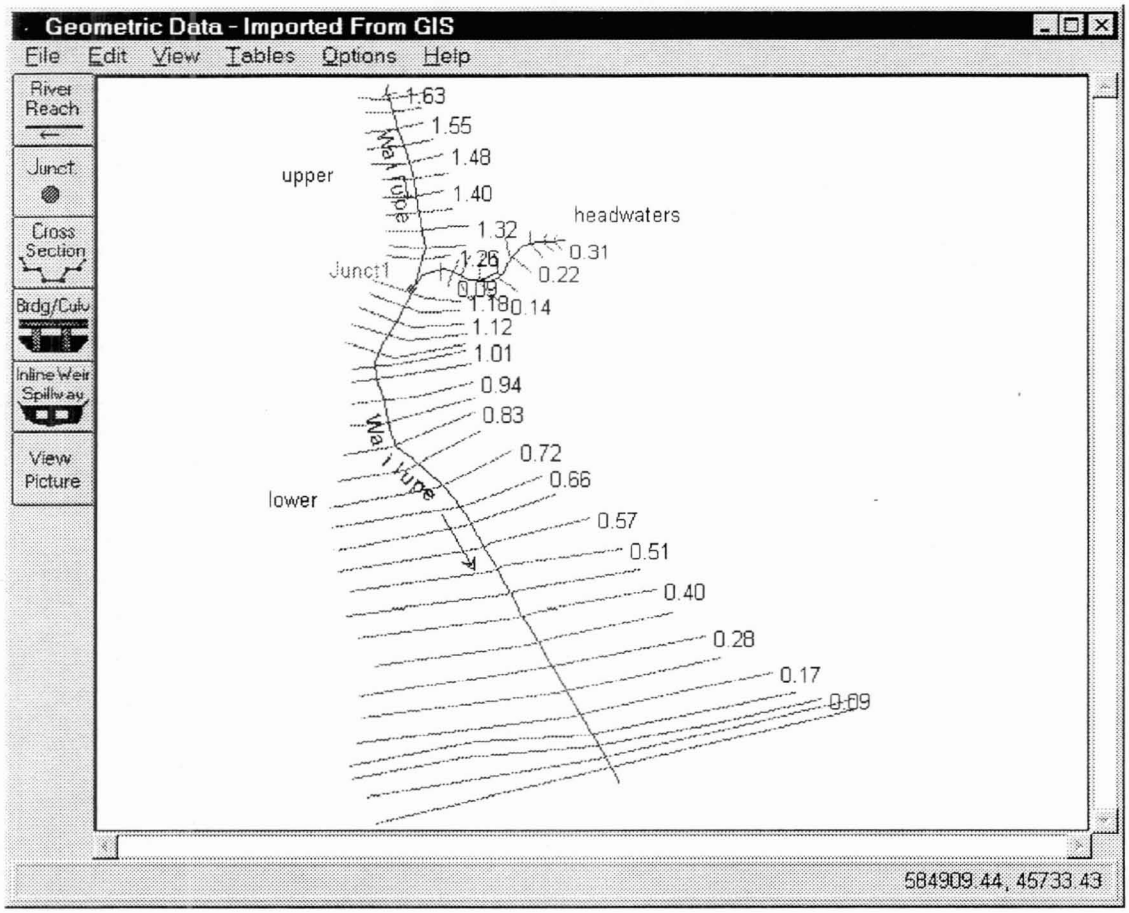


Figure 13.2 River System Schematic of Imported GIS Data

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. At a minimum, the user is currently required to enter Manning's n values for all of the cross sections, and junction reach lengths if there are any junctions in the data set. 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).

Importing Additional Geometry From The GIS

The user also has the option of getting additional cross-section data from the GIS. Additional cross sections can be extracted from the GIS and written to the GIS text file for import into HEC-RAS. When this is done, if the original cross section data as well as the new cross sections are contained in the GIS file, the HEC-RAS software will ask the user if they want to completely clear the existing geometry file and start a new one or if they want to update currently opened geometry data. If the user chooses to update the currently opened geometry data and if the GIS import file contains cross sections that are already in the geometry file, the program will ask the user to select one of the following three options **pertaining to the duplicate cross sections only**:

Clear existing XS and replace with new data. This option completely eliminates the existing cross sections and replaces it with the new data found in the GIS import file.

Update existing cross section with new data only. This option does not clear the existing cross section, but it does update the cross section with any of the data found in the GIS import file. Information like the ground points, main channel bank stations, and downstream reach lengths will be updated. Manning's n values, and optional cross section properties will not be changed from the original data.

Do not modify existing cross section data. This option tells the program not to modify anything about the existing cross section data, and to only add any new cross sections that are found in the GIS import file.

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 13.3.

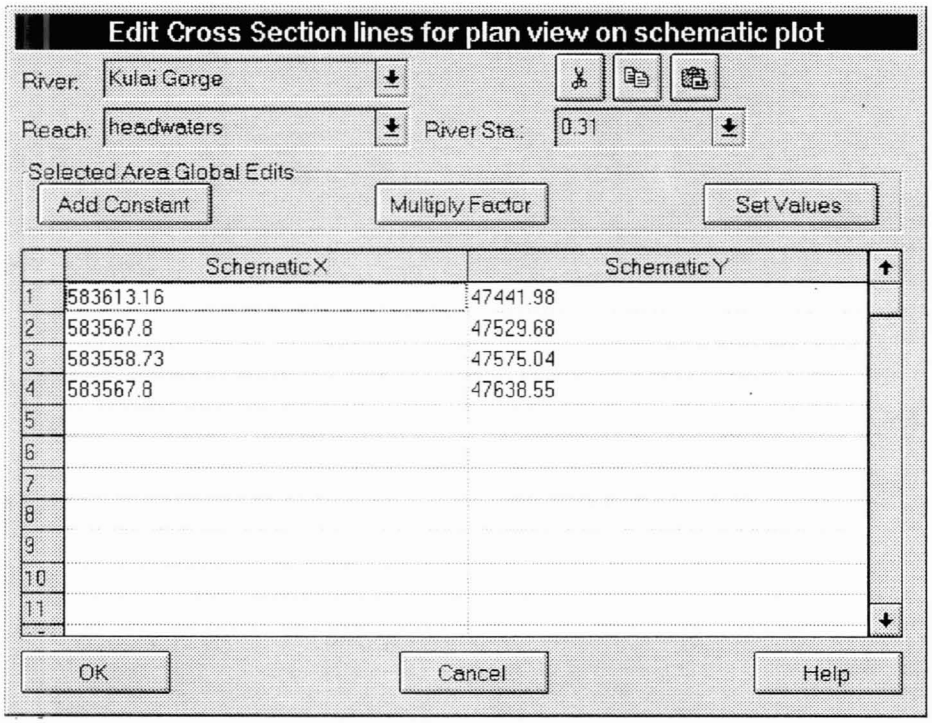


Figure 13.3 Cross Section Schematic Line X and Y Coordinates

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 13.4.

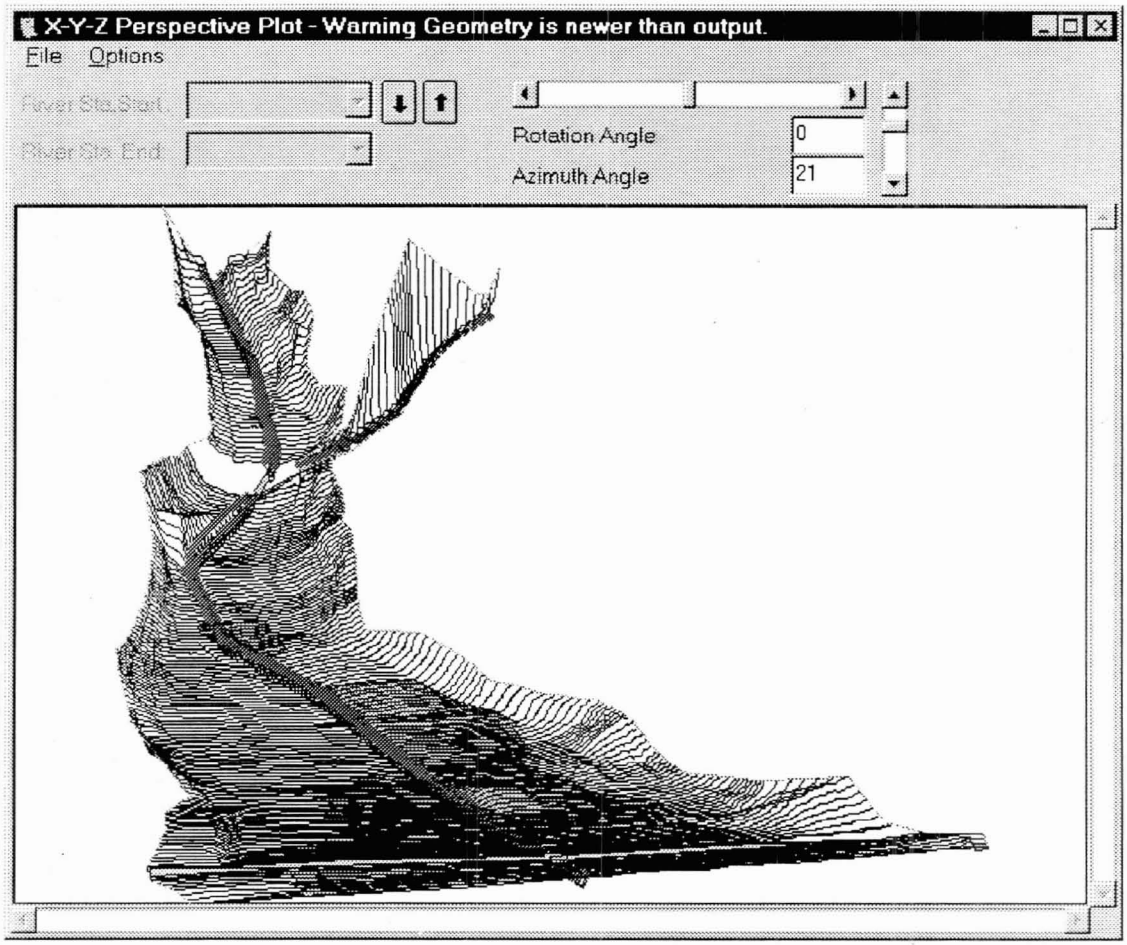


Figure 13.4 XYZ Perspective Plot With GIS Data

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 13.5.

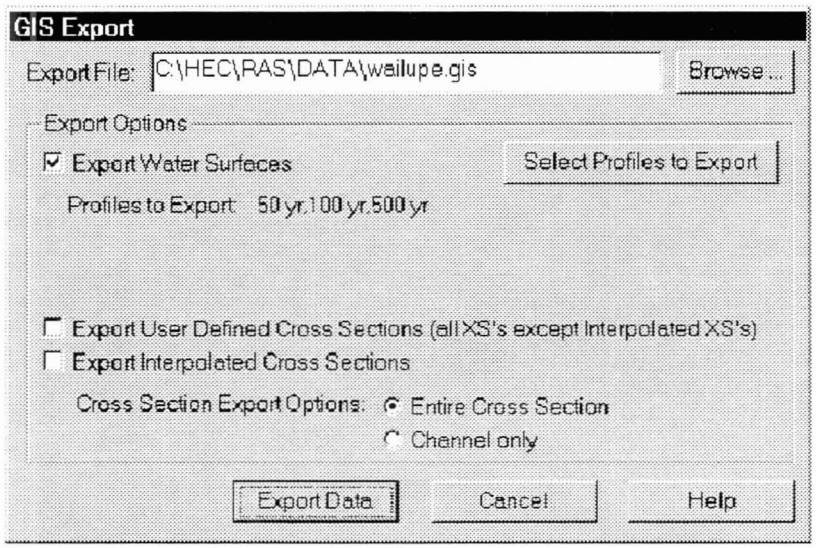


Figure 13.5 Window to Enter a Filename For Exporting Results to GIS

As shown in Figure 13.5, 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.

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.

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..

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..

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.

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.

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.

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.

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, containing reach locations and connectivity.
3. A descriptions 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:
```

minimum import file 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
       lines omitted
      478144.53, 64296.61, 123.72, 22.41
  END:
END STREAM NETWORK:
```

minimum import stream network section

A reach endpoint is represented by a record containing the keyword "ENDPOINT:" followed by four comma-delimited fields containing the endpoint's 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-section 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-section 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. Stationing is given in miles for data sets with plane units of feet and in kilometers for data sets with plane units of meters. A cut line is composed of the label "CUT LINE:" followed by an array of 2D locations. A cross-section 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).
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 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. Profile names can contain up to 16 characters. They must begin with a letter, and must not contain blanks.

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 susequent 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 terrrain, 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. Stationing is given in miles for projects using English units and in kilometers for projects using metric units.

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

```
BEGIN HEADER:

  DTM TYPE: TIN
  DTM: /HEC63/USR1/EVANS/WAILUPE/WAI_TIN (TIN)
  STREAM LAYER: /HEC63/USR1/EVANS/WAILUPE/WAI_STR
  NUMBER OF REACHES: 3
  CROSS-SECTION LAYER: /HEC63/USR1/EVANS/WAILUPE/WAI_XS
  MAP PROJECTION: STATEPLANE
  PROJECTION ZONE: 5101
  DATUM: NAD27
  UNITS: ENGLISH

END HEADER:

BEGIN STREAM NETWORK:

  ENDPOINT:   582090.19,    49360.46,   220.17, 1
  ENDPOINT:   583638.69,    47559.38,   266.80, 2
  ENDPOINT:   582307.31,    46985.66,   112.84, 3
  ENDPOINT:   584128.44,    41274.97,    -3.41, 4

  REACH:
    STREAM ID: Kulai Gorge
    REACH ID: Headwaters
    FROM POINT: 2
    TO POINT: 3
    CENTERLINE:
      583638.69,    47559.38,  266.80,    0.33
  11 lines omitted
      582307.31,    46985.66,  112.84,    0.00
  END:

  REACH:
    STREAM ID: Wailupe
    REACH ID: Upper
    FROM POINT: 1
    TO POINT: 3
    CENTERLINE:
      582090.19,    49360.46,  220.17,    1.65
  14 lines omitted
      582307.31,    46985.66,  112.84,    1.19
  END:

  REACH:
    STREAM ID: Wailupe
    REACH ID: Lower
    FROM POINT: 3
    TO POINT: 4
    CENTERLINE:
      582307.31,     46985.66,   112.84,   1.19
  33 lines omitted
```

```
      584128.44,      41274.97,          -3.41,       0.00
  END:

END STREAM NETWORK:

BEGIN CROSS-SECTIONS:

  CROSS-SECTION:
    STREAM ID: Kulai
    REACH ID: Headwaters
    STATION: 0.312
    BANK POSITIONS:  0.5562,  0.6294
    REACH LENGTHS:   84.541,  89.110,  82.013
    CUT LINE:
      583613.16,      47441.98
      583567.80,      47529.68
      583558.73,      47575.04
      583567.80,      47638.55
    SURFACE LINE:
      583613.16,      47441.98,       309.69
  29 lines omitted
      583567.80,     47638.55,       278.10
  END:

6 Cross-Sections omitted

  CROSS-SECTION:
    STREAM ID: Kulai
    REACH ID: Headwaters
    STATION: 0.019
    BANK POSITIONS:  0.4454,  0.4799
    REACH LENGTHS:   187.942, 193.195, 163.246
    CUT LINE:
      582769.62,      46950.81
      582598.07,      46978.64
      581981.41,      47224.45
    SURFACE LINE:
      582769.62,     46950.81,       167.62
  78 lines omitted
      581981.41,     47224.45,       169.89
  END:

  CROSS-SECTION:
    STREAM ID: Wailupe
    REACH ID: Upper
    STATION: 1.629
    BANK POSITIONS:  0.4781,  0.5615
    REACH LENGTHS:   55.965,  53.626,  40.370
    CUT LINE:
      582159.78,      49259.60
      582013.48,      49223.76
    SURFACE LINE:
      582159.78,     49259.60,       235.49
  29 lines omitted
```

```
  END:
```

7 Cross-Sections omitted

```
    STREAM ID: Wailupe
    REACH ID: Lower

    BANK POSITIONS:  0.5236,  0.5686
    REACH LENGTHS:   171.000, 164.796, 159.249

      582723.17,     46846.45
      582426.44,     46878.92

    SURFACE LINE:
      582723.17,    46846.45,      161.92

      581953.51,    47082.99,      165.01
  END:



  CROSS-SECTION:
    STREAM ID: Wailupe
    REACH ID: Lower
    STATION: 0.037
    BANK POSITIONS:  0.5034,  0.5155
    REACH LENGTHS:   82.365,  192.982, 137.742
    CUT LINE:
      586214.12,     42127.92
      581980.99,     40806.06
    SURFACE LINE:
      586214.12,    42127.92,        4.01
```

71 lines omitted

```
      581980.99,    40806.06,        6.39
  END:

END CROSS-SECTIONS:

FILE COMPLETE: 28 October 1996,  17:17
```

Sample HEC-RAS Geographic Data Export File

```
BEGIN HEADER:

# RAS output file created 29 Oct 96 16:36:41 Tuesday
# by HEC-RAS
  NUMBER OF REACHES: 3
  NUMBER OF CROSS-SECTIONS: 46
  NUMBER OF PROFILES: 2
  MAP PROJECTION: STATEPLANE
  PROJECTION ZONE: 5876
  DATUM: NAD27
  PLANE UNITS: FEET
  ELEVATION UNITS: FEET
  STATION UNITS: MILES
  PROFILE NAMES: PF#1, PF#2

END HEADER:

BEGIN CROSS-SECTIONS:

  CROSS-SECTION:
    WATER ELEVATIONS: 265.8189, 268.7436
    CUT LINE:
      583613.16,   47441.98
      583567.80,   47529.68
      583558.73,   47575.04
      583567.80,   47638.55
  END:

  CROSS-SECTION:
    WATER ELEVATIONS: 257.7377, 270.6435
    CUT LINE:
      583537.56,   47429.88
      583474.05,   47529.68
      583474.05,   47568.99
      583492.20,   47662.74
  END:
```

119 cross-sections omitted

```
  CROSS-SECTION:
    WATER ELEVATIONS: 5.964032, 6.43543
    CUT LINE:
      586202.53,   42188.21
      583941.76,   41483.75
      581932.30,   40961.43
  END:

  CROSS-SECTION:
    WATER ELEVATIONS: 5.662289, 6.12576
    CUT LINE:
      586214.12,   42127.92
```

```
      581980.99,  40806.06
  END:

END CROSS-SECTIONS:

BEGIN BOUNDS:

  PROFILE LIMITS:
    PROFILE ID:PF#1
    POLYGON:
      582013.48 , 49223.76
    63 lines omitted
      581819.14 , 49209.56

    POLYGON:
      581953.51 , 47082.99
    141 lines omitted
      581934.965 , 47008.78

    POLYGON:
      583567.8 , 47638.55
    43 lines omitted
      583530 , 47650.645
  END:

  PROFILE LIMITS:
    PROFILE ID:PF#2
    POLYGON
      582013.48 , 49223.76
    63 lines omitted
      581819.14 , 49209.56

    POLYGON:
      581953.51 , 47082.99
    141 lines omitted
      581934.965 , 47008.78

    POLYGON:
      583567.8 , 47638.55
    43 lines omitted
      583530 , 47650.645
  END:

END BOUNDS:

#FILE COMPLETE: 17 Jan 97 16:38:04 Friday
```

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.
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 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 Inv El Dn	ft	Culvert inside invert elevation downstream.
Culv Inv El Up	ft	Culvert inside invert elevation upstream.
Culv Nml Depth	ft	Normal depth for this culvert (and flow).
Culv Q	cfs	Flow through all barrels in a culvert group.
Culv Vel In	ft/s	Velocity inside of culvert at inlet.
Culv Vel Out	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. 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 Invert	ft	Gate spillway invert elevation.
Gate Open Ht	ft	Height of gate opening.
Gate Q Total	cfs	Flow through all gate openings in a gate group.
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 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.
Ineff El Left	ft	The elevation of the left ineffective area.
Ineff El Right	ft	The elevation of the right ineffective 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.
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 Top Rd	ft	Elevation where weir flow begins.
Momen. EG	ft	Energy grade elevation upstream of bridge for momentum method.
Momen. WS	ft	Water surface elevation upstream of bridge for momentum method.
Num Trials	#	Current number (or final number) of trials attempted before the energy equation is balanced.
Obs WS	ft	Observed water surface elevation.
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.
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 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 Right	cfs	Flow in right overbank.
Q Total	cfs	Total flow in cross section.
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 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 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.
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. $SF = A_T Y_{cent} + (Q^2\beta)/(gA_{act})$
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 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.
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. Elev	ft	Calculated water surface from energy equation.
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 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 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.

HEC-RAS
Version 2.2, September 1998
Release Notes

Since the last release of the HEC-RAS software (Version 2.1, October 1997) we have found and fixed several bugs. Additionally, a few new features have been added to this version.

Important Note: Version 2.2 of HEC-RAS will only run on computers using Windows 95, 98 or Windows NT 4.0. This version of HEC-RAS will not run under the Windows 3.1 or 3.11 operating systems. This is due to the fact that Microsoft no longer supports Windows 3.1 or 3.11. All of the Microsoft software development products that we use no longer allow us to develop a version of HEC-RAS that will run under Windows 3.1 or 3.11. If you are using Windows 3.1 or 3.11, you must either upgrade your operating system to Windows 95, 98 or NT 4.0, or you can continue to use HEC-RAS 2.0, noting the bugs in that version.

New Features and Modifications

1. HEC-RAS now has the ability to model floating ice as well as dynamic ice jams. This option is available from the geometric data editor and is documented in all three of the HEC-RAS manuals.

2. Reading data from and writing results to HEC-DSS (HEC Data Storage System). The user can read flow and stage data from HEC-DSS and use it for computing water surface profiles. Reading Data from HEC-DSS is accomplished from the Steady Flow Data editor. The user must first establish connections between HEC-RAS cross section locations and the DSS file pathnames. This is accomplished by using the "Set Locations for DSS Connections" option from the File menu of the Steady Flow Data editor. Once these connections are established, the user can import data by using the "DSS Import" option, also from the file menu of the Steady Flow Data editor. Additionally, user's can write computed water surface profiles to HEC-DSS. The export of results to DSS is accomplished under the File menu of the main HEC-RAS window.

3. An option to view data contained in an HEC-DSS file was also added. This option allows the user to graphically plot or tabulate data contained in DSS. This option is available from the View menu on the main HEC-RAS window.

4. Internal Rating Curves. The user can now put a rating curve at any cross section. During the computations, the program will take the water surface from the rating curve instead of computing a value. Internal Rating curves are added from the Steady Flow Data editor.

5. The number of profiles allowed has been increased from 15 to 100.

6. GIS Export now allows users to export the cross section surface lines to the GIS file. Additionally, the user can specify to send just the main channel only or the entire cross section. This feature was added to allow users to supplement their terrain data with surveyed cross section information.

7. Picture viewer now shows the picture filename in the window title and a copy to clipboard button was added.

8. Boundary Condition Editor - When a boundary condition in the table is double clicked, the appropriate dialog editor is opened

9. Graphics - Zooming out has changed meaning. The "Zoom Out" option now doubles the zoom view in height and width. A new option called "Full Plot" was added. When this option is selected, the entire plot is displayed. Also, a "Pan" option was added for moving the graphic around when zoomed in.

10. The user can now rotate the three dimensional graphic while zoomed in on a piece of the river system. The old version would automatically zoom out when attempting this.

11. In previous versions, the path to the RAS data directory could not have any spaces in it. This restriction has been removed. Also, the path length had a set limit. This problem has also been fixed.

12. Two new methods of specifying the distance for cross section interpolation were added. A constant distance interval from the upstream cross-section and a set specific location from the upstream section.

13. Internal bridge cross section editor has added options to insert a row and to delete a row.

14. An option was added to the schematic editor to reverse the stationing data of the cross section geometry. This was added for those data sets that have the cross sections entered from right to left looking downstream, instead of from left to right.

User Interface Bugs Found in Version 2.1 and Fixed for Version 2.2

1. Error plotting critical depth in profile plots with metric data sets.

2. The simulation option of computing critical depth using the secant method to search for multiple critical depths option was not being saved in the data file.

3. Printing XS tables - The column widths were not set correctly, this made the text from one column run over into the next cell.

4. Report Generator - The reporting of the bridge computation options of Momentum Class B defaults and Pressure Criteria were incorrect.

5. Manually adding GIS reach lines would not move a junction. Now the junction and the first point in all reaches move with the new reach location.

6. The filename and path to an HEC-RAS project could not contain any spaces in earlier versions. This limitation has been removed.

7. Error renaming river stations. In 2.1 there is an error renaming the river stations from the cross section editor. The work around is to copy the station to the desired river station, then delete the old location, or use the River Station table from the geometric editor.

8. Rating curves were not correctly converted from Metric to English. This will affect running Metric data as well, since all computations are in English units. The flow was converted as if it was a length, not a volume per time.

9. Channel Modification. The graphic on the Channel Modification window did not show that the blocked obstructions will still be in place after the cuts. The graphic was modified so that the blocked obstructions show they still reach down to the cut.

10. Cross Section plot (and Rating curve) would show "12312" in the mouse tracking box. This would stay there until you moved the mouse over the plot. This box is now empty when you open the plot window.

11. The misspelled word in the tool tip for the cross section button on the geometric data editor has been corrected.

12. Removing junctions from the geometry caused some problems if the lower reach of the junction to be deleted had a junction on its downstream end.

13. Bridge Scour with SI units.

14. The abutment scout using Froehlich's equation with SI units produced incorrect scour results.

15. If the pier size fraction D50 and D90 were not entered the program would crash when computing scour depths.

16. GIS exporting water surface. If you use encroachments on your data set, then the bounding polygon for the first profile (Base) was incorrectly computed. This will affect the ability to generate the inundated area in the GIS package. The encroached profiles worked fine. This is a problem for data sets with encroachments only.

17. GIS cross section cut lines. If the GIS cut lines were edited when bridges, culverts, or inline weirs were present in the geometry, a message would always display that told users that the GIS cross section cut lines would not be displayed unless all XS's had GIS cut lines. This message was display even if all XS had GIS cut line data.

18. The title on Geometry Schematic window did not change when the user changed the geometry file on the steady flow analysis window.

19. SI Data Sets. Bridges with internal cross sections specified were not converted correctly before SNET computations.

20. Inline Weir Gate Editor. When the broad crested weir option was selected, the weir coefficient text field disappeared.

21. Pipe arch culverts. In SI units, the span was not computed when a rise was entered.

22. Bridge Culvert Editor. If a bridge culvert picture was zoomed in and the user edited the data on a sub editor (such as the pier editor) the bridge picture would update and show the new data but the scaling would be missing or incomplete on the plot.

23. Geometric Schematic did not print at a specified scale when it was set by the user.

24. User defined tables were not printed by the report generator.

25. Generating River Stations from the geometric table option to rename river stations returns an incorrect message when bridges and culverts are present.

26. When increasing the number of profiles, the boundary condition for the last profile, was supposed to be used for subsequent profiles. There was an error in that it would use the first profiles, not the last.

27. A few bridge clipping problems were fixed.

28. Converting project units back and forth caused some of the plan files to have an incorrect geometry file selected.

29. Report Generator - Rating curve boundary conditions were not being printed as part of the flow data.

30. Metric problem - General profile plot (i.e. plotting velocity in profile) was not converting all the data to metric correctly. This was fixed.

Solutions to some graphics problems

1. A user reported that he was having errors in zooming in on graphics. He found that when he downloaded a new device driver for his ATI3DRage2Plus video card that it worked.

Bugs Found in Version 2.1 of Computation Program SNET

1. If a hydraulic jump occurred in the cross section downstream of a bridge, then the reach length was not being calculated correctly. This caused the profile plot to be displayed incorrectly.

2. The program cannot calculate a supercritical profile through two or more nonidentical culverts. When this is attempted, the program is supposed to skip the culvert computations. This check was not being performed correctly and the program was occasionally crashing.

3. The program was sometimes reporting the pressure/weir solution as the final answer even when it didn't converge (it did, however, report a warning). When the pressure/weir solution does not converge, the program now defaults to the energy method answer.

4. If a culvert crossing had weir flow and the culvert inlet control answer was higher than the outlet control answer, but both the culvert inlet and outlet are submerged (hence this causes the inlet answer to be discarded), the final result was not being reported correctly (a work around was to set the culvert to "outlet only"). This has been fixed.

5. Encroachments at a cross section just upstream of a junction were not being performed correctly if the junction was being modeled with the momentum method.

6. A levee at a bridge node could occasionally cause the program to crash.

7. An encroachment that stopped at a vertical wall that was going up (from outside to inside) would cause the program to crash.

8. For the class B momentum solution, the program was not always correctly deciding which interior bridge cross section to set to critical depth.

9. Several modifications have been made to the momentum solution that may cause differences in the final water surface solution. The force from the weight of water and the force from the road embankment have been changed and should be more hydraulically correct. The static force exerted by the upstream pier was being calculated from the wrong water surface. The downstream pier was using the wrong internal bridge section. Both of these have been corrected.

10. A bug was found that was keeping the pressure/weir flow solution from converging in certain cases. This has been corrected and the pressure/weir solution should now be more stable.

11. A single opening bridge that is completely outside of the channel stations could, on rare occasion, cause the program to crash.

12. Low and high arch culverts had a bug that could sometimes result in an incorrect answer being reported and it could, at other times, cause SNET to crash.

13. The Yarnell bridge method is not considered valid if the water hits the low chord or overtops the road. However, the program was not always disregarding the Yarnell method when this happened.

14. The program was having a problem performing encroachments at multiple opening river crossings. If the encroachment would block out the entire opening, the program would blow-up. This has been fixed. Now if the encroachment blocks the entire opening, that opening is automatically turned off.

15. There was a problem in the WSPRO bridge methodology. Occasionally the program was having trouble computing the discharge coefficient.

16. An inline weir that has gates but no weir flow would sometimes fail to converge. This has been fixed.

17. Occasionally bridges with pier debris were getting incorrect calculations of area and wetted perimeter inside of the bridge at the upstream end.

18. There was a problem with cross sections that had more than 5 blocked ineffective flow areas. The problem was in the calculation of critical depth.

19. There was a problem with encroachments at inline weirs and gated spillways. The program was making the correct calculations, but was outputting the wrong surcharge just upstream of the weir/spillway.

20. Occasionally a culvert that was under inlet control and had weir flow, was not balancing to a final solution within the specified tolerance.