

**Some Basics of**

**HEC-RAS**

**River Analysis System**

**August 2006**

# Table of Contents

- Downloading and Installing HEC-RAS.....1**
  
- Purpose of HEC-RAS .....1**
  
- Beginning a New Project .....2**
  - Creating a Reach/Entering a Junction .....3**
  - Adding Cross Section/Junction Data.....7**
  - Adding Flow/Boundary Conditions .....15**
  - Saving the Data .....18**
  - Computing the RUN (steady/unsteady, subcritical/supercritical/mixed) ...19**
  - Viewing the Results .....20**
  
- Error Messages, Warnings, and Notes .....23**
  
- Interpolating Cross Sections .....24**

This manual is intended to aid the reader in performing many of the most commonly used features of the HEC-RAS program quickly and with no prior knowledge of how the program operates. All of the features of the HEC-RAS program are covered in more detail in the program *User's Manual* and the *Hydraulic Reference Manual* which may be downloaded from the following site:

<http://www.hec.usace.army.mil/software/hecras/hecras-document.html>

## **Downloading and Installing HEC-RAS**

As of August 2006 the most current version of HEC-RAS is 3.1.3, which may be downloaded from the following site: <http://www.hec.usace.army.mil/software/hecras/hecras-download.html> (you may find this site quickly by typing “*hec-ras download*” into Google.) The file size is 39.4MB and may be downloaded using either high or low bandwidth; follow the directions at this site for downloading and running the program.

## **Purpose of HEC-RAS**

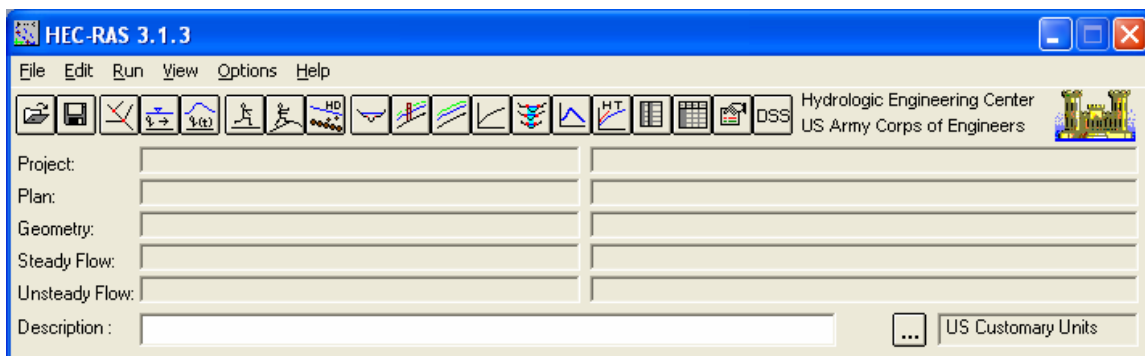
One of the major purposes for using HEC-RAS is to compute water surface elevations. This is where Hydrology and Hydraulics come together; hydrology allows us to calculate information such as 100-year flood flow, probable maximum flood flow (PMF), etc., then using this information in HEC-RAS we can figure out what the possible hydraulic conditions will be for a given study area. Therefore, knowing the water surface elevation under various flow conditions can be helpful for many reasons, some of which are: to evaluate possible flooding; for bridge/culvert design work; for riprap placement; to

determine construction risk; for obtaining permits from natural resource agencies; when building structures such as bridges, culverts, homes, erosion control measures, etc., to evaluate the differences in water surface elevation before and after construction in order to comply with local, county, and FEMA regulations. HEC-RAS may also be used to generate flow velocities for use in studying erosion and scour or for obtaining permits.

It is good to keep in mind that although a model will never be 100% correct, it may still be useful. Greater confidence in a model may be gained through calibration, comparing information collected on a particular project with corresponding data computed by the model.

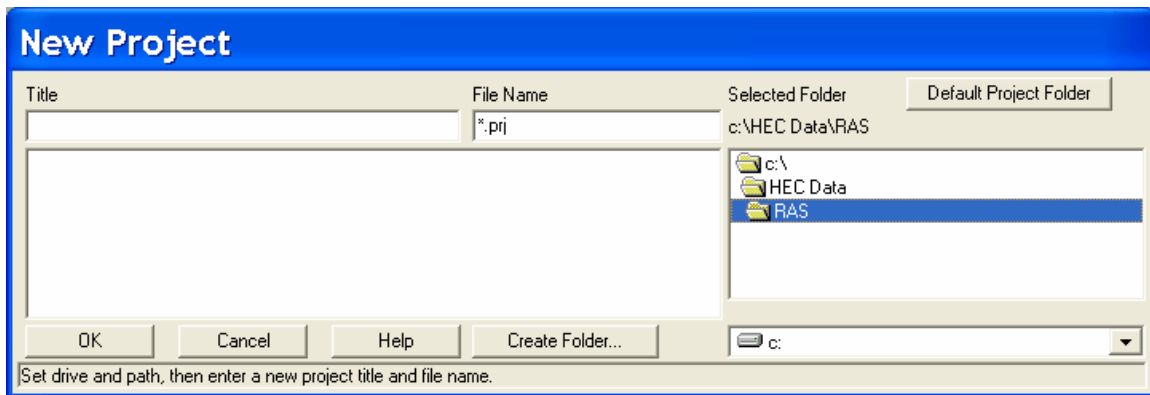
## Beginning a New Project

To start the HEC-RAS program, double-click on the **HEC-RAS 3.1.3** icon located on your desktop, or go to the **Start** menu and click on all **programs, HEC, HEC-RAS, HEC-RAS 3.1.3**. When the program opens, the main HEC-RAS window will pop up on your desktop:



Many of the program features may be selected using the File, Edit, Run, or View menus, or by using the list of buttons underneath these menus.

Begin a new project by selecting **File**, and **New Project**. The New Project window will appear:



Next select the drive (e.g., c:\) from the drop down box in the lower right hand corner and path (e.g., HEC) found on the right, in which to save work completed on the project.


Then enter a project title (e.g., Sample) and a file name (e.g., Prob1.prj) and click **OK**. A file name must always contain the “.prj” file extension. A message box with the new project title and directory will appear as a confirmation. If it is correct, click **OK**, if not, click **Cancel** and complete the process again.

The next step is to select the unit system (English or Metric) that you would like to use for the entire project by going to the main HEC-RAS window and clicking on the **Options** menu, then select **Unit System** and choose the desired system, then click **OK**. For the following examples, make sure that **US Customary** is selected.

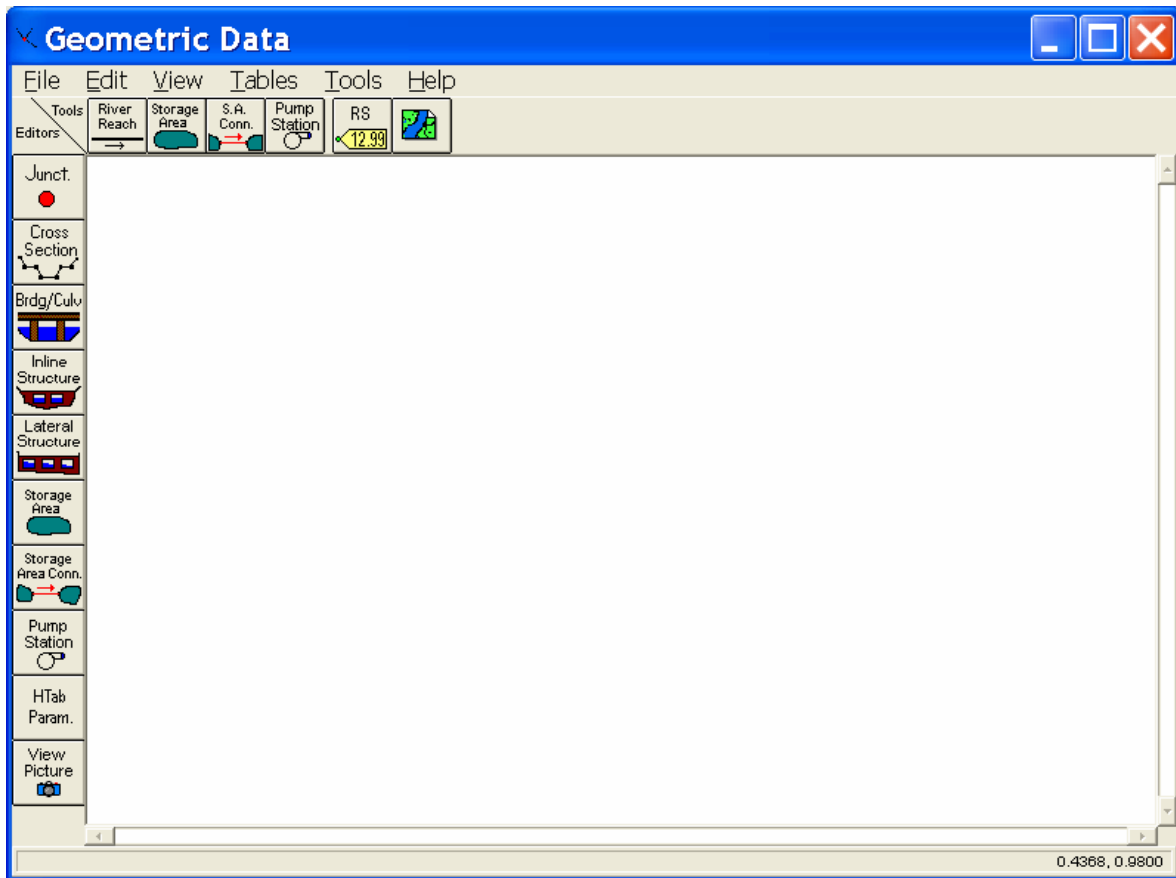
## Creating a Reach/Entering a Junction

Creating a Reach and entering a Junction means drawing a schematic of the system to be modeled including, as will be shown in the following example, a junction where a tributary meets a river, dividing the river into an upper reach and a lower reach.

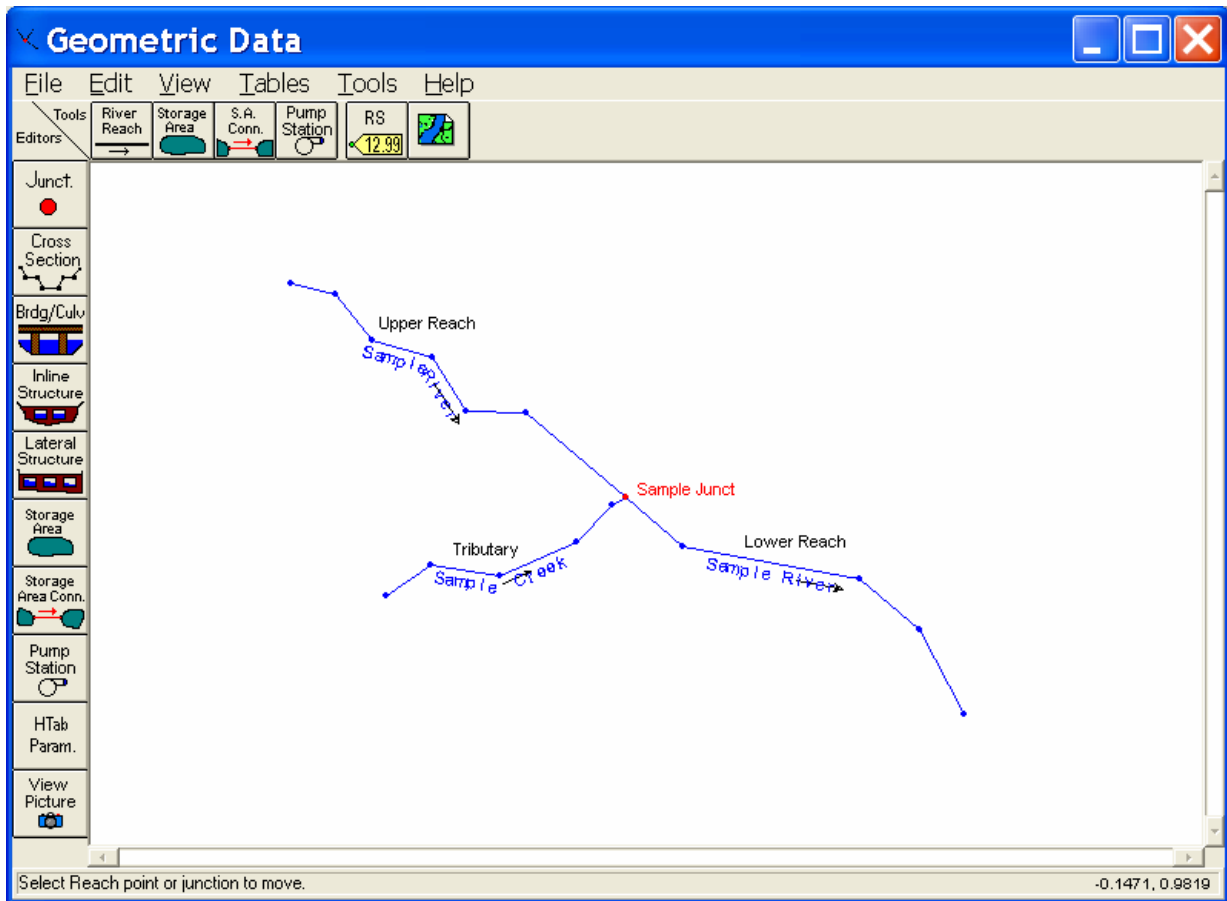
Begin either by clicking on the **Edit** menu and selecting **Geometric Data**, or by

clicking on the Geometric Data button: 

The following Geometric Data window will appear:



Use the following example to learn how to create a river system schematic; it may be helpful to read through the entire process before beginning your schematic:



To create this river system, first draw the entire collection of line segments for “Sample River” by clicking the **River Reach** button. Click in the white drawing space to begin drawing the reach, left-click to draw each line segment of the river, and double-click to finish drawing the river reach. It is not necessary to draw the schematic in perfect detail as this has no affect on how the program computes the results, in fact, you may represent the system using only a couple of straight lines if you desire. You are then prompted to enter the “River” and “Reach” identifiers as shown:



In this case “**Sample River**” and “**Upper Reach**” are entered knowing that you will include a tributary and a junction later.

The next step is to draw the tributary by using the same method for drawing the first river reach; end the tributary by double-clicking where it connects with “Sample River.” The following box will appear:



For the river name enter “**Sample Creek,**” and for the reach name enter “**Tributary,**” then click **OK**.

Another message appears that asks if you want to split “Sample River” river on reach “Upper Reach.” Click **Yes** and a new box appears to prompt you to enter a new reach name below the split in the river as shown:





Enter “**Lower Reach**” and click **OK**. You are then prompted to enter a name for the junction where the two reaches meet. Enter “**Sample Junct**” and click **OK**.

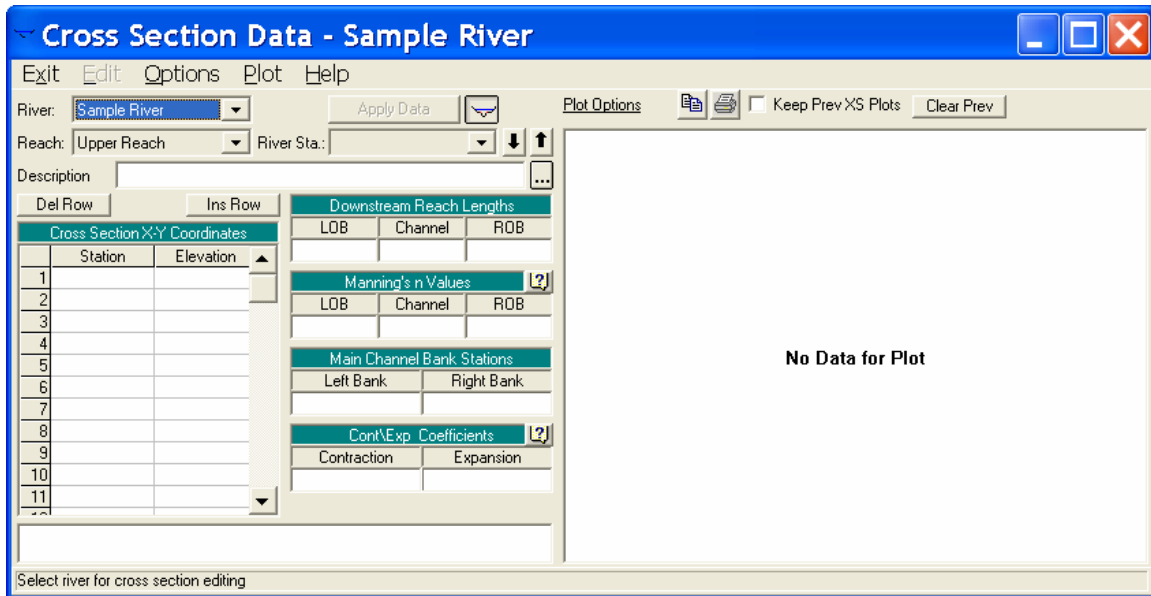
You may continue adding reaches (tributaries) and junctions using the process above. You may also modify the existing river system by selecting from a list of choices under the **Edit** menu of the Geometric Data window. If you make a mistake, you may choose the Edit menu and select the option that allows you to delete a reach; then delete any or all of the reaches that you may need to. It is a very good idea to continually save your data. To save the geometric data, click the **File** menu on the Geometric Data window, then click **Save Geometry Data** and enter a geometry data title such as “**Base Geometry Data.**”

## **Adding Cross Section/Junction Data**

After drawing the schematic of the system, you must enter the geometric data (i.e., cross section and junction data) which the program will use to complete the computations for water surface elevations, etc.

To enter the cross section data, go to the Geometric Data window as was done to complete the river schematic. You do this by clicking on the **Edit** menu from the main

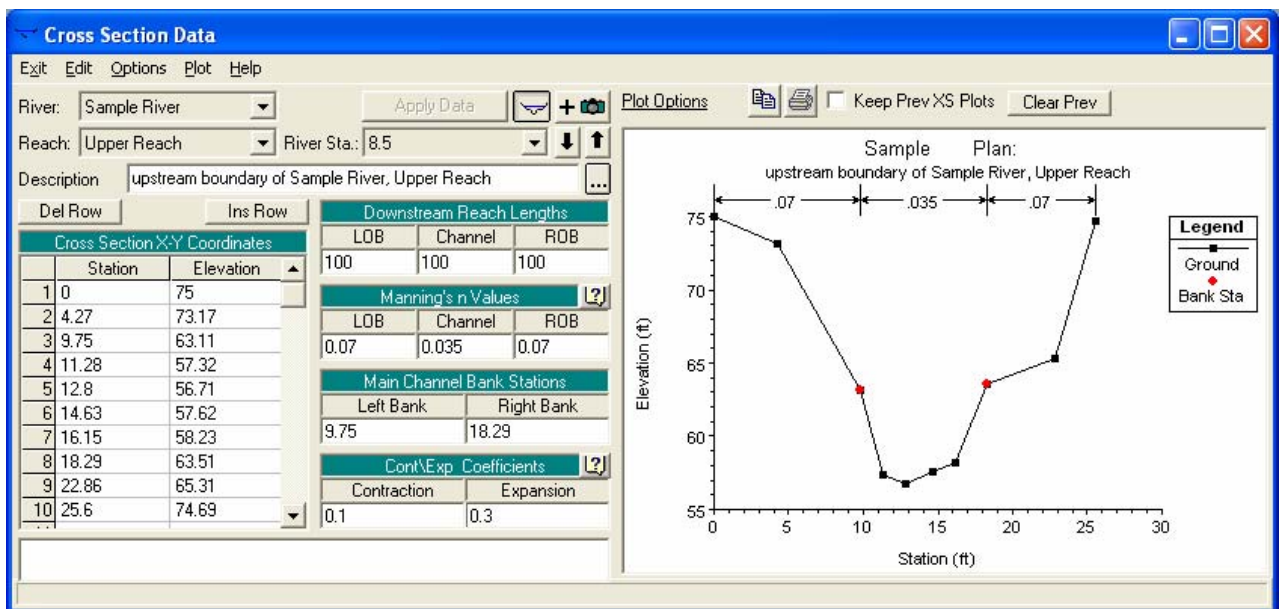
HEC-RAS window, and then click on **Geometric Data**. From the Geometric Data window, click on the **Cross Section** button. Clicking this button will bring up the following window:



The next step is to select a **River** and **Reach** (shown in the upper left hand corner). For this example, **Sample River** and **Upper Reach** are selected to work with. In order to enter cross section data you must select the **Options** menu from the Cross Section Data window, and select **Add a new Cross Section**. A small window will pop up asking you to enter a new river station for the new cross section in reach “Upper Reach.” Although the number you enter for a river station does not have to represent an actual river station, it must be a numeric value which will identify this cross section and its location in relation to all other cross sections in this reach. Cross sections are located from upstream (the river station with the highest numeric value) to downstream (the river station with the lowest numeric value). For this example, the first river station is entered as **8.5** (representing river mile 8.5 along Sample River). For this example all numbers are in US customary units. The next step is to enter a short description to help you identify this

cross section from among the other cross sections later. Type in **upstream boundary of Sample River, Upper Reach**.


Now we can begin to enter the cross section data (station and elevation data, or X-Y coordinates). This data is entered from the left bank of the cross section to the right bank of the cross section (the left side of the cross section will have lower numbers than the right side) as though you were looking in the downstream direction, or in the direction of flow. For this example, enter the numbers as shown in the following figure:



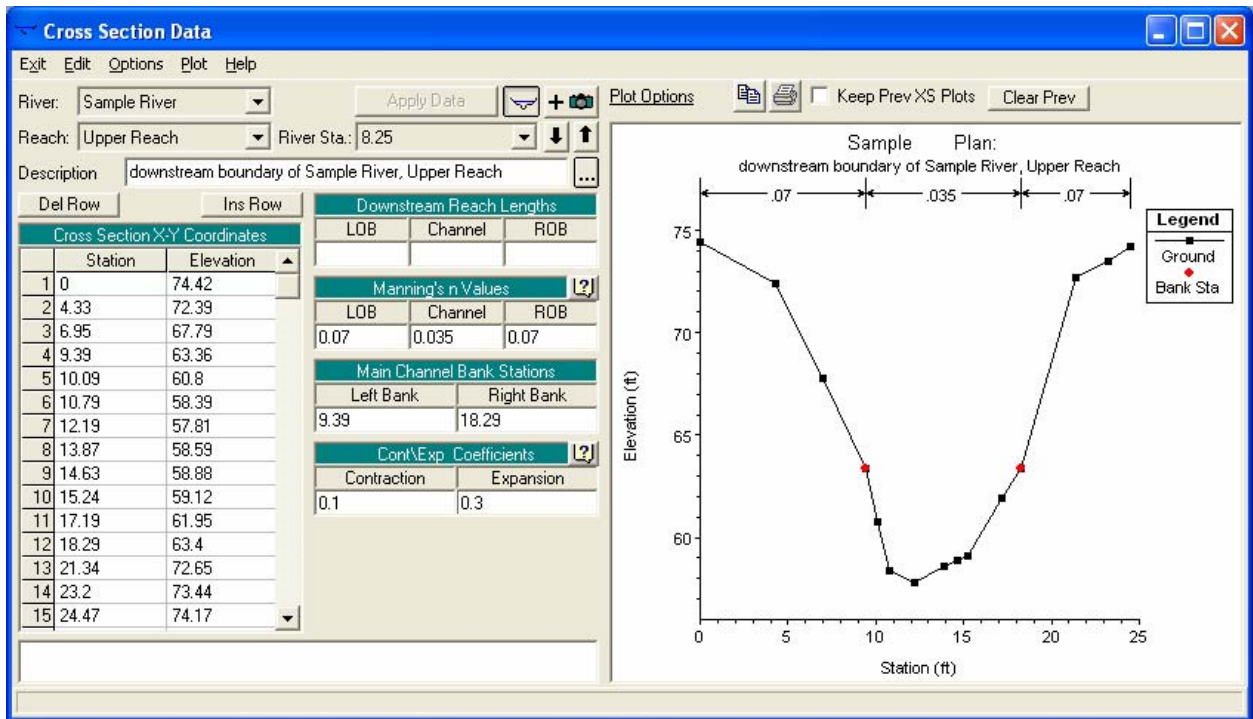
Click the **Apply Data** button (top center) when finished to generate a cross section plot.

If this plot matches your cross section data, then you are ready to enter the rest of the information in the boxes shown just to the right of where you entered the Station/Elevation data. Next, enter the **Downstream Reach Lengths** (left over bank, main channel, and right over bank), which is the length from this cross section to the next downstream cross section. Enter the corresponding **Mannings's n Values** (for a description of these roughness values, click the small question mark to the right of the Manning's n Values title block). Next, enter the **Main Channel Bank Stations**. The

Main Channel Bank Stations delineate the left and right over bank areas (flood plains) from the main channel and will show up as red dots on the cross section plot. The Contraction and Expansion coefficients are used to compute the energy losses associated with the contraction and expansion of flow in a system. For most projects, 0.1 and 0.3 are used, respectively. When you are finished entering the data as shown in the previous figure, click the **Apply Data** button.

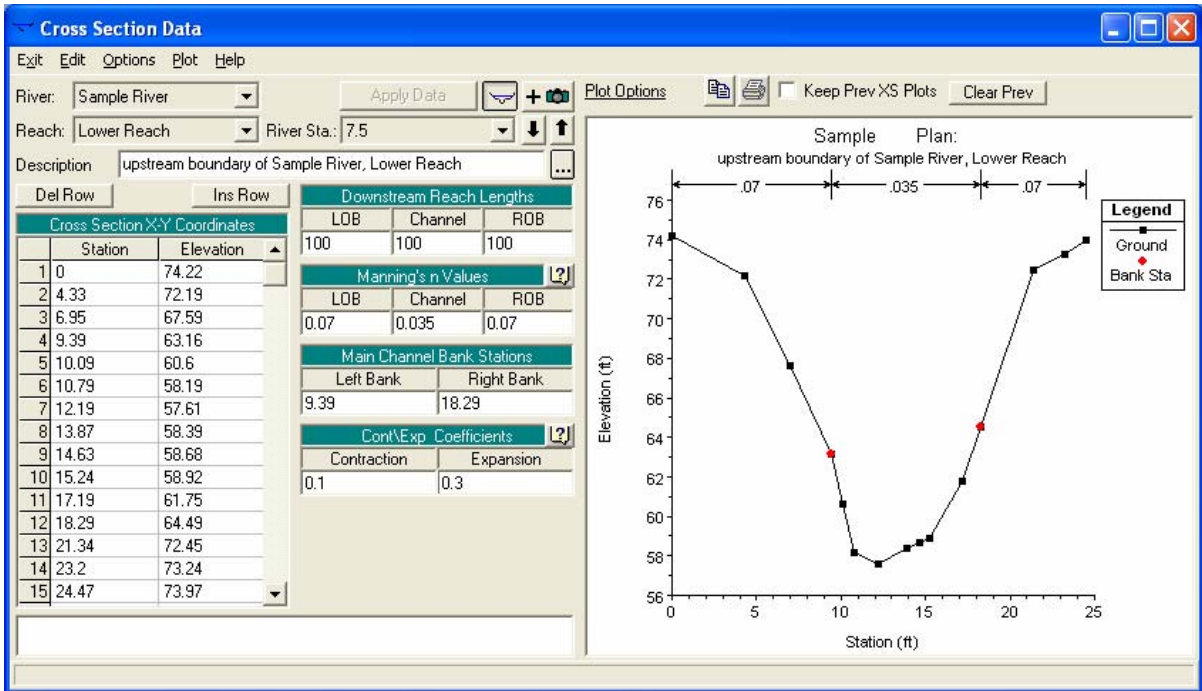
A file containing a picture may also be linked to a particular cross section by clicking on the camera button:  choosing the appropriate River, Reach, and River Station, and then by clicking on the **Add Picture** button to search for and add the corresponding picture file.

There are other useful options for entering and editing data found by clicking the **Options** menu of the Cross Section Data window. We will use the Copy Current Cross Section option to create the next cross section. Copying an existing cross section will simply copy all of the existing data exactly as it is, you must then modify the copied cross section data as needed (e.g., river station number, station/elevation data, etc.). The advantage of this option is that it allows you to interpolate a new cross section, or to copy an existing cross section which can be modified quickly using choices listed in the Options menu. Select **Copy Current Cross Section** and enter the next river station which is **8.25**, then click **OK**. For this example, modify all of the cross section information to match the following figure, adding station/elevation data as necessary:



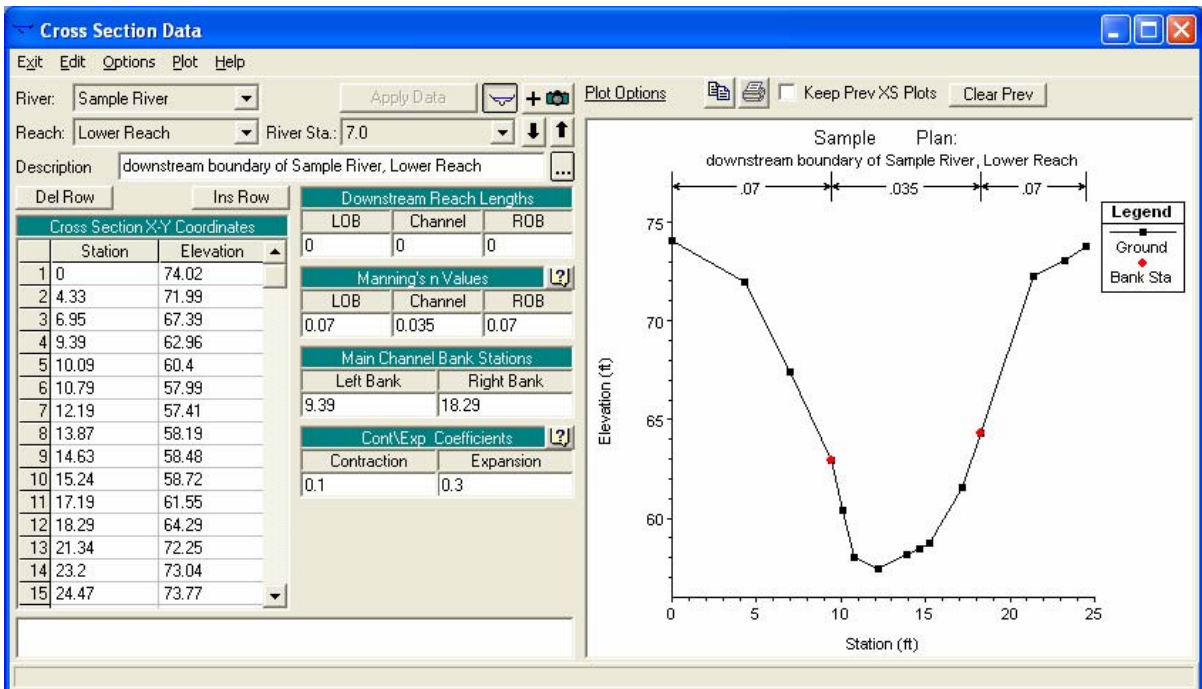
The Downstream Reach Lengths are left blank in this case because the length between the upstream and downstream reach includes the junction where the tributary (Sample Creek) flows into “Sample River,” and this length will be entered later as Junction Data. Click the **Apply Data** button when finished to generate the cross section plot.

We must now enter upstream and downstream boundary cross section information for the “Lower Reach” of Sample River and for “Sample Creek.” First, select **Sample River** from the River drop down box (upper left hand corner) on the Cross Section Data window and **Lower Reach** from the Reach drop down box. Next click on the **Options** menu and choose the **Add a new Cross Section** option. For this example, enter **7.5** as the new river station and click **OK**. Type in **upstream boundary of Sample River, Lower Reach** as the description. Next, enter the following cross section information as shown:



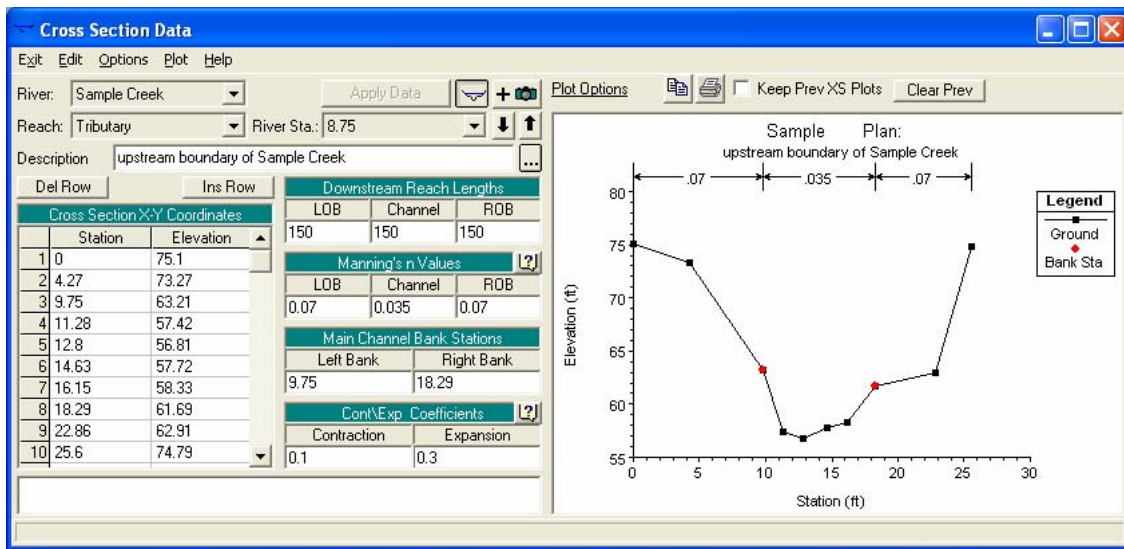
Click the **Apply Data** button when finished.

Repeat the previous procedure to add a cross section (you may use the Copy Current Cross Section option) at the end of the **Lower Reach** of **Sample River** and use the following river station, description, and cross section information as shown:

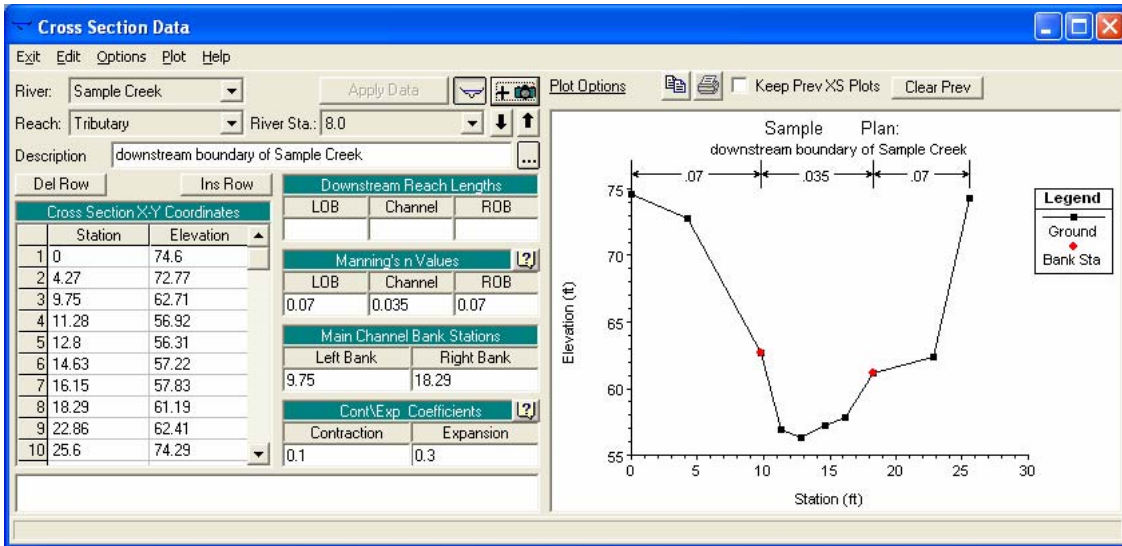


Since this is the downstream boundary of Sample River, the Downstream Reach Lengths are entered as zero or they may be left blank. If you choose the Copy Current Cross Section option, you may more easily duplicate the next cross section for this example by modifying the existing elevations. This is done by clicking **Options, Adjust Elevations**, type **-0.2**, and click **OK**.

Next, enter the following information for the upstream cross section of the “Tributary” of “Sample Creek”:




Click the **Apply Data** button when finished, and then enter the following information for the downstream cross section of “Sample Creek”:



Click the **Apply Data** button when finished.

The final step in this example is to enter the junction data. This is done by first

going to the Geometric Data window and clicking on the **Junction** button: 

The Junction Data window will appear; enter the following information to indicate the lengths across each section of the junction:

The 'Junction Data' window shows the following configuration:

- Junction Name: Sample Junct
- Computation Mode:  Energy,  Momentum
- Add Friction,  Add Weight

Length across Junction	Junction Length (ft)	Tributary Angle (Deg)
From: Sample River - Lower Reach		
To: Sample River - Upper Reach	20	
To: Sample Creek - Tributary	30	

The Energy computation mode is already selected which means that the energy equation will be used to model the junction. If the Momentum computation mode (momentum




equation) is used, the angle at which the tributary enters will have to be entered. Finally, save the geometric data by clicking **File** on the Geometric Data window, and then choose **Save Geometry Data** to update the geometric data saved previously (see page 7).

## **Adding Flow/Boundary Conditions**

The amount of flow through the system must be entered along with certain boundary conditions before running the program to compute the desired results. The amount of flow through a system will depend on the type of study conducted. Determining which boundary conditions are required depends on the conditions of the system and the type of model being run. The options for running the model are steady and unsteady flow analysis, and within each of these are options for modeling a subcritical, supercritical, or mixed flow regime. As stated in the HEC-RAS *Hydraulic Reference Manual*, “subcritical profiles computed by the program are constrained to critical depth or above, and supercritical profiles are constrained to critical depth or below. In cases where the flow regime will pass from subcritical to supercritical, or supercritical to subcritical, the program should be run in a mixed flow regime mode.” For this example we will perform a **steady flow** analysis using a **subcritical** flow regime.

Begin by going to the main HEC-RAS window, click on **Edit** and then **Steady**

**Flow Data**, or click on the steady flow data button: 

This will bring up the following steady flow data window:

**Steady Flow Data** [min] [max] [close]

File Options Help

Enter/Edit Number of Profiles (2000 max):

Locations of Flow Data Changes

River:

Reach:   River Sta.:

Flow Change Location				Profile Names and Flow Rates
	River	Reach	RS	PF 1
1	Sample Creek	Tributary	8.75	
2	Sample River	Upper Reach	8.5	
3	Sample River	Lower Reach	7.5	

Edit Steady flow data for the profiles (cfs)

The next step is to enter the amount of flow at each location (Sample Creek, the Upper Reach of Sample River, and the Lower Reach of Sample River). Enter the following flow information for each location as shown in the white boxes under PF 1:

**Steady Flow Data - Flow 01** [min] [max] [close]

File Options Help

Enter/Edit Number of Profiles (2000 max):

Locations of Flow Data Changes

River:

Reach:   River Sta.:

Flow Change Location				Profile Names and Flow Rates
	River	Reach	RS	PF 1
1	Sample Creek	Tributary	8.75	500
2	Sample River	Upper Reach	8.5	1500
3	Sample River	Lower Reach	7.5	1500

Select river for adding a new flow change location.

Click the **Apply Data** button (upper right hand corner) when finished. The heading PF 1 may be changed to represent the event that this amount of flow represents (e.g., 100yr, or PMF, etc.). Also, more flow profiles may be added by increasing the number of profiles to allow you to model more than one event at a time.

Next, we need to enter the necessary reach boundary conditions in order to establish the starting water surface at the upstream and downstream ends of the system and enable the program to begin calculations. For a subcritical flow regime, boundary conditions are only necessary at the downstream end; for a supercritical flow regime, boundary conditions are only necessary at the upstream end; and for the mixed flow regime, boundary conditions are necessary for both the upstream and downstream ends of the system. Boundary conditions at junctions are considered internal and will already be listed in the steady flow boundary conditions table. Click on the **Reach Boundary Conditions** button on the steady flow data window to view the Steady Flow Boundary Conditions table:

**Steady Flow Boundary Conditions**

Set boundary for all profiles
  Set boundary for one profile at a time

Available External Boundary Condition Types

Selected Boundary Condition Locations and Types

River	Reach	Profile	Upstream	Downstream
Sample Creek	Tributary	all		Junction=Sample Junct
Sample River	Upper Reach	all		Junction=Sample Junct
Sample River	Lower Reach	all	Junction=Sample Junct	Junction=Sample Junct

Select Boundary condition for the downstream side of selected reach.

The five options for entering boundary conditions are: know water surface elevations; critical depth (when selected, the program will calculate critical depth for each of the profiles and use that as the boundary condition); normal depth (an energy slope must be entered which is used to calculate the normal depth at that location using Manning's equation); rating curve (an "elevation versus flow" rating curve must be entered from which the program interpolates the water surface elevation for each profile given the flow).

For this example, click on the **Downstream** box for the "Lower Reach" of "Sample River," shown highlighted in the previous figure. Click the **Normal Depth** button (top center) and enter **0.02** as the energy slope, then click **OK**. Click the **OK** button of the Steady Flow Boundary Conditions window, and you will be brought back to the Steady Flow Data window. Click the **Apply Data** button (top right hand corner) to allow the program to accept all the input data. The final step is to save the flow/boundary conditions data. From the Steady Flow Data window, click **File**, then click **Save Flow Data** and enter a title such as "**Base Flow Data**," and click **OK**. This completes the process for entering the flow/boundary conditions data.

## **Saving the Data**


At this point, we have indicated how to save the different files of a HEC-RAS project as we have covered each section. The complete list of files that can make up a HEC-RAS project are: 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

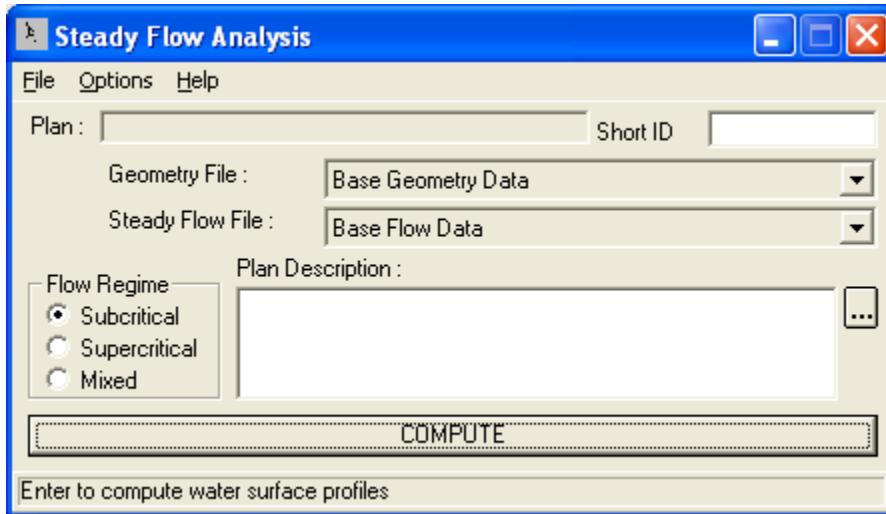
(.F01 to .F99); one file for each set of unsteady flow data (.U01 to .U99); one file for each set of sediment data (.S01 to .S99); and one file for each set of hydraulic design data (.H01 to .H99).

It is always a good idea to save a file after entering the data for that particular section (e.g., the steady flow data, geometry data, etc.), and more importantly to save a copy of the existing model before making modifications (e.g., to flow amounts, boundary conditions, etc.) and before you run a model. Files are saved by clicking **File** from the corresponding data window (i.e., the Geometry Data window, Steady Flow Data window, etc.) and clicking on the **Save Data** option for that particular file.

### **Computing the RUN (steady/unsteady, subcritical/supercritical/mixed)**

In order to compute a run you must first enter all of the geometry data, flow data, etc. related to the project. The next step in computing our steady flow analysis is to define a Plan. If there are multiple files for a project, the plan defines, for instance, which geometry and flow data are to be used for the run and allows you to enter a description and short identifier to quickly access the correct output files and other run information later on.

To create a plan for this example, go to the main HEC-RAS window and click on **Run**, and then click **Steady Flow Analysis**, or click the steady flow analysis button:  to open the Steady Flow Analysis window:



Since we only have one geometry and steady flow file, we may leave these files as they appear in the Steady Flow Analysis window. To create a new plan, click **File**, and then click **New Plan**, and enter a name for the plan such as “**Sample Plan**,” and click **OK**. You are then asked to enter a short identifier for this particular run, enter “**Sample Run**,” and click **OK**. When you are conducting multiple runs, it is important to enter a short identifier that will help you to quickly locate a particular plan when viewing multiple plan output from the graphics and tables. You may also type in a description of the plan in the space provided. Since the Subcritical Flow Regime is already selected, click **File** and then **Save Plan**. Finally, to perform the steady flow calculations, click the **COMPUTE** button and allow the program to compute the results of the run, and then click the **Close** button on the pop-up window.

## Viewing the Results

After computations have finished successfully, you have several options for viewing output, they are: cross section plot; profile plot; general profile plot; rating curve; X-Y-Z perspective plot; detailed tabular output at a specific cross section (a cross section

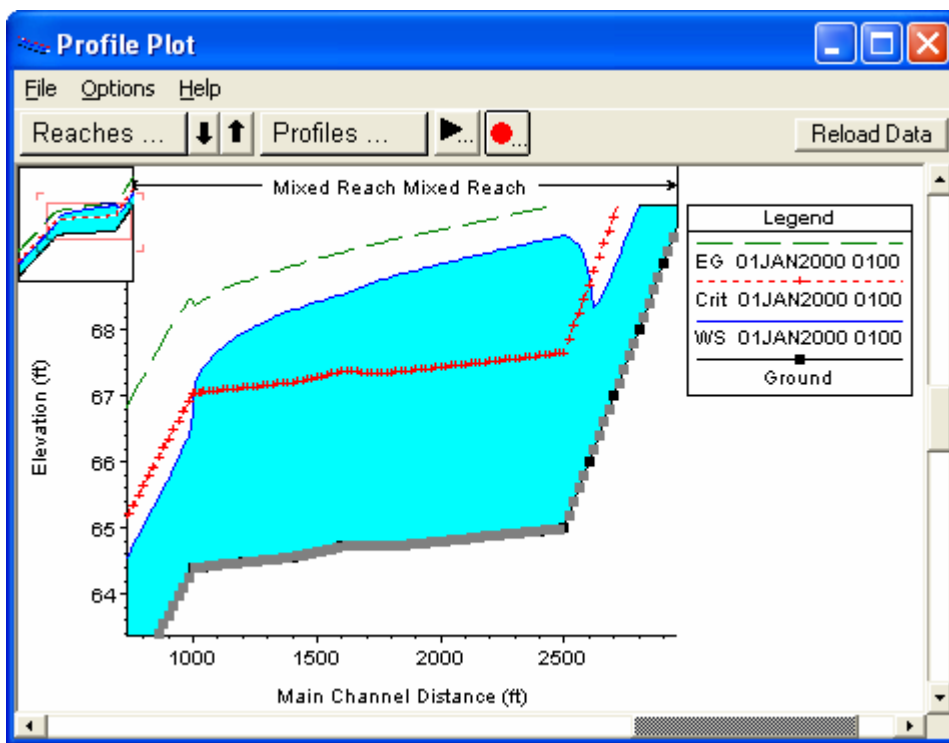
table); and a limited tabular output of many cross sections (a profile table). To access these output options, go to the main HEC-RAS window and click **View**, then choose one of the viewing options, or click on one of the following buttons located on the main HEC-RAS window:



It may be helpful to note that the cross section plot is looking from left to right in the downstream direction (direction of flow), and the profile plot shows flow direction from right to left. Use the appropriate buttons and drop down boxes on the profile windows to cycle between rivers/reaches, and to include/exclude certain rivers/reaches in a plot. Several types of profile tables may be used to display specific information, including at hydraulic structures (e.g., bridges and culverts). These are accessed by going to the main HEC-RAS window and clicking **View**, then **Profile Summary Table**, and then clicking on the **Std. Tables** menu. By choosing the **Options** menu from the Profile Summary Table, you may configure the table to include any of the information you desire by clicking on **Define Table**. When viewing the results of a run using tables, be sure to check the Options menu of the tables (i.e., Detailed Output, Profile Summary) in order to view the information you seek. Depending on the study you are conducting, different output options will help you find and present the results you need. Tables and graphics may also be sent directly to a printer or to the Windows Clipboard to add in a report. To select these options, click either **Print** or **Copy to Clipboard** on the **File** menu of the displayed table or graphic. Another method for pasting tables or graphics into a Word document is to copy it as a screen shot. This allows you to copy and paste a table exactly as it looks in the HEC-RAS program. Make sure that the “Table” or “Graphic” is the

active window (the last window that you opened), press and hold the **Alt** button, then press the **Print Screen** button (on your keyboard) to copy the table, next paste the table or graphic into the Word document.

The following is an example (not a part of our sample project) of interpreting the results of a run using a profile plot. It may be very helpful to look at a profile plot to determine areas of concern. If you were running a subcritical flow regime, you could look for potential areas of concern on the profile plot by searching for the location(s) where the water surface passes above and below the critical depth, which may indicate the location of a hydraulic jump. The following profile plot is an example of this:



At the right side of the profile plot, a hydraulic jump occurs. This is indicated by the blue water surface line (flow direction is from right to left) which starts out below the red critical depth line and then quickly jumps above it. Over on the left side of the plot, the water passes from subcritical flow to supercritical flow (water surface begins above the



red critical depth line and then passes underneath it). The green line indicates the energy grade line.

## **Error Messages, Warnings, and Notes**

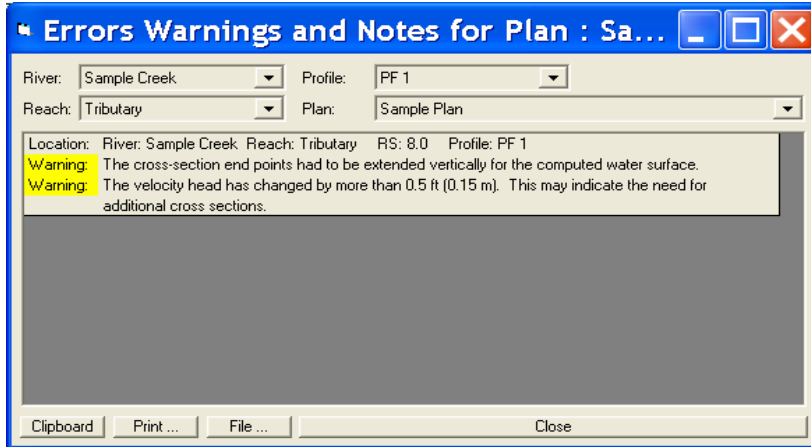
After computing a run, you should always check if the program has generated any error messages or warnings. These messages may help you to figure out how to get the program to complete a run or suggest ways by which you can help the program to generate more accurate results. Whenever a warning message is generated, you should review the hydraulic results at that location to make sure they are reasonable, and if they are then the message may be ignored. If the results are not reasonable, you will probably need to take action to make the warning message disappear from future runs.

To view the complete table of error messages and warnings, go to the main HEC-RAS window and click the **View** menu, and then click on **Summary Err, Warn, Notes**,

or click on the appropriate button: 

Many warning messages are caused by bad or inadequate data. Common problems that cause warning messages to occur are: cross sections are spaced too far apart; the starting and ending stations of cross sections are not high enough (the computed water surface is higher than either end point of a cross section); a bad starting water surface elevation (a boundary condition is specified that is not possible for the specified flow regime); bad cross section data can cause many problems (most often the program cannot balance the energy equation and will default to critical depth); notes (these messages provide information about how the program is performing the computations).

From the example we completed, the following table will appear:



For “Sample Creek,” we can see that the computed water surface started higher than either of the end points of the cross section at river station 8.0. Also, because the energy change due to velocity (velocity head) changed more than the program default (0.5ft), the program indicates that more cross sections may be needed for Sample Creek.

By choosing “Sample River” and “(All Reaches),” to include both the upper and lower reaches, from the drop down boxes (upper left hand corner), warnings indicate a need for additional cross sections for both the upper and lower reaches as well as a message indicating that the normal depth at river station 7.0 on the lower reach was set to critical depth because the slope is too steep.

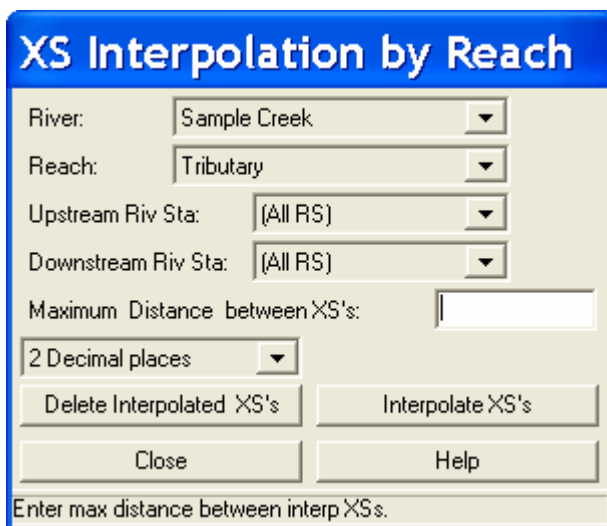
## Interpolating Cross Sections

Additional cross sections are often needed to adequately model friction losses throughout a system. A common warning message indicates the need of additional cross sections because the change in velocity head is too large. There are three ways to interpolate cross sections in HEC-RAS. The first is to copy an existing cross section and then adjust the station and elevation data, the cross section data editor allows you to raise

or lower elevations and to shrink or expand portions of any cross section. The second and third methods allow you to automatically interpolate cross section data.

Using our previous example, we will interpolate cross sections for all of the reaches (those on Sample River and Sample Creek). To begin interpolating cross sections, go to the Geometric Data window and click on the **Tools** menu option. Move your cursor over **XS Interpolation** and click on **Within a Reach**. You will notice that you also have the option to interpolate cross sections between two selected existing cross sections.

The following window appears:



Choose “**(All Rivers)**” from the first drop down box. Type in **20** as the Maximum Distance between XS’s, and click the **Interpolate XS’s** button. Next click the **Close** button and you will see the interpolated cross sections on the system schematic shown on the Geometric Data window; each interpolated cross section has an asterisk (\*) next to it. As you review the warnings table after re-computing the run, you will notice that the warning messages which indicated a need for additional cross sections have mostly disappeared.