

Creating a simple 1D HEC-RAS Model for a Single Reach

Prepared by

Venkatesh Merwade

School of Civil Engineering, Purdue University

vmerwade@purdue.edu

April 2020

Introduction

The objective of this exercise is to learn basic functions in HEC-RAS to create a 1D steady state model. It is expected that you are familiar with 1D St. Venant's, energy, and Manning's equations. Although you will not need these equations, but they are used behind the scenes. If you want to get into details of HEC-RAS that are not covered in this tutorial please refer to the HEC-RAS users manual.

Pre-requisite: Watch the Introduction to 1D Hydraulic Modeling video at:

<https://www.youtube.com/watch?v=o9bwunC4G7A>

Learning Outcomes

- Creating a simple 1D HEC-RAS model/project
- Creating a single reach geometry for HEC-RAS
- Running 1D steady state simulation in HEC-RAS
- Run HEC-RAS simulation
- Visualize HEC-RAS results

Student Time Required: 50 mins

Computer Requirements

You must have a computer with windows operating system with HEC-RAS 5.0.x installed. You can download HEC-RAS and related documents for free from the US Army Corps of Engineers Hydrologic Engineering Center website

<https://www.hec.usace.army.mil/software/hecras/>

Data Requirement

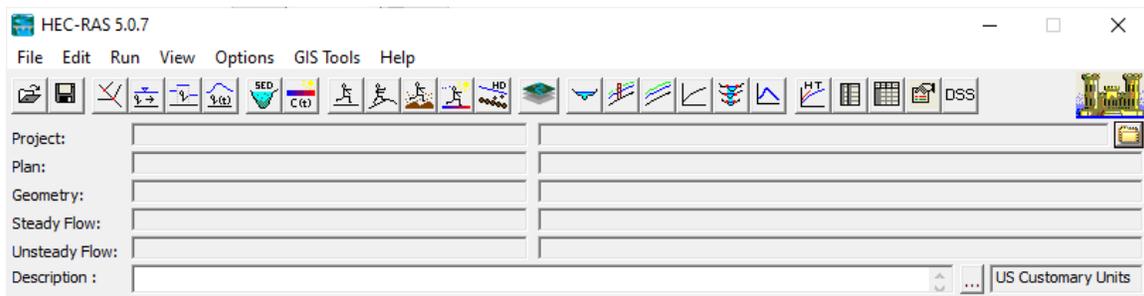
None.

Instructions

Start HEC-RAS

In most typical installations on windows system, you go to Start→Programs→HEC→HEC-RAS 5.0.x. This path may be different in computer labs. Just search for HEC-RAS and open the program.

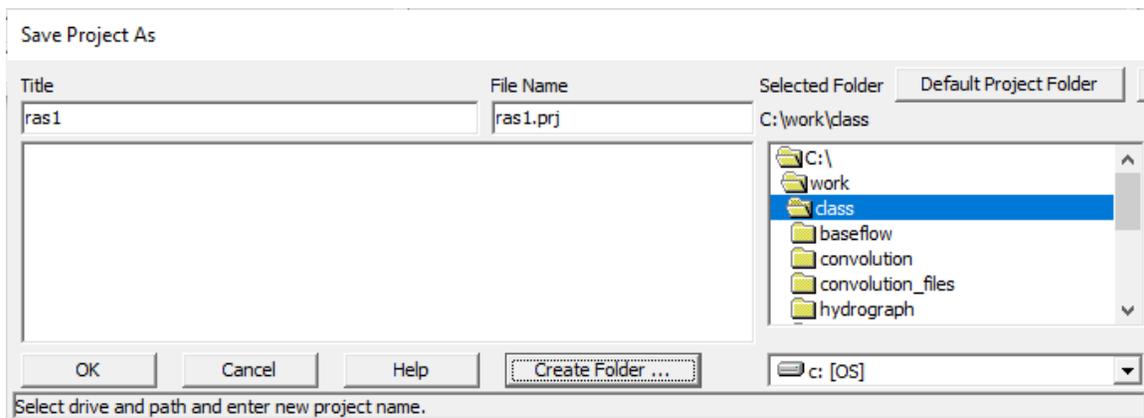
HEC-RAS Interface



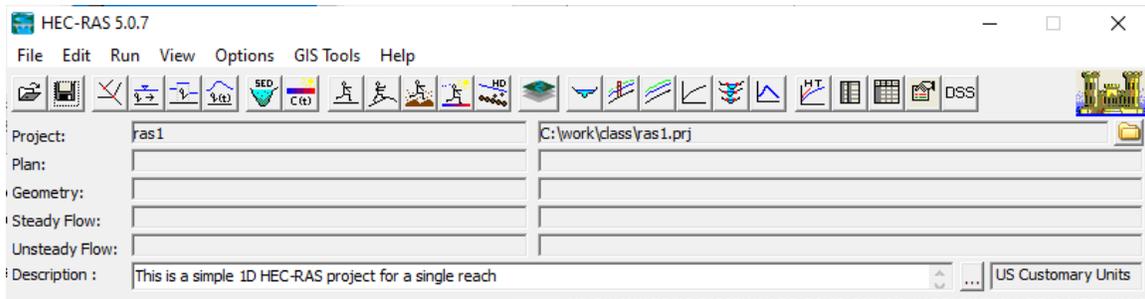
The interface consists of a list of menu (File, Edit, Run, etc.) on the top, followed by a list of tools (which are a subset of few key items from the menu) and a list of files (Project, Plan, Geometry, Steady Flow and Unsteady Flow). The description box (bottom) can be filled with whatever description you want for the project. By default, the units are US Customary, which can be change b going to Options in the Menu bar and changing the Unit System.

Creating and Saving HEC-RAS Project

Save the project by going to File→Save Project as.. name the project as ras1 in a folder of your choice. If needed, you can use the Create Folder option to create a new folder and save the project inside that new folder.

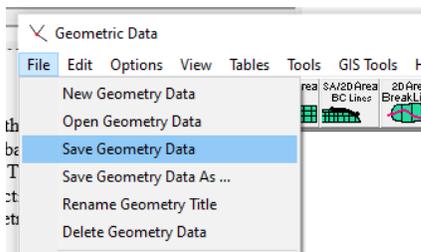


After the project is saved, you will see that the Project box in the list of files gets filed with ras1 and its location is also displayed on the interface. A HEC-RAS project file is saved with .prj extension (note: a GIS projection file also has .prj extension, but both files are very different!). Lets also go ahead and provide some description for the Description box.



Creating and Saving Geometry File

Once we create the project, the next step is to define the river geometry. A 1D river geometry file describes the shape of a river using, at bare minimum, the river centerline, cross-sections (location and shape) and bank points. The geometry file will also include information on Manning's n values for each cross-section. Go to **Edit**→**Geometry Data**.. to open the Geometry Data Editor. Within the Geometry Data editor, go to **file** and save it using the **Save Geometry Data** option.



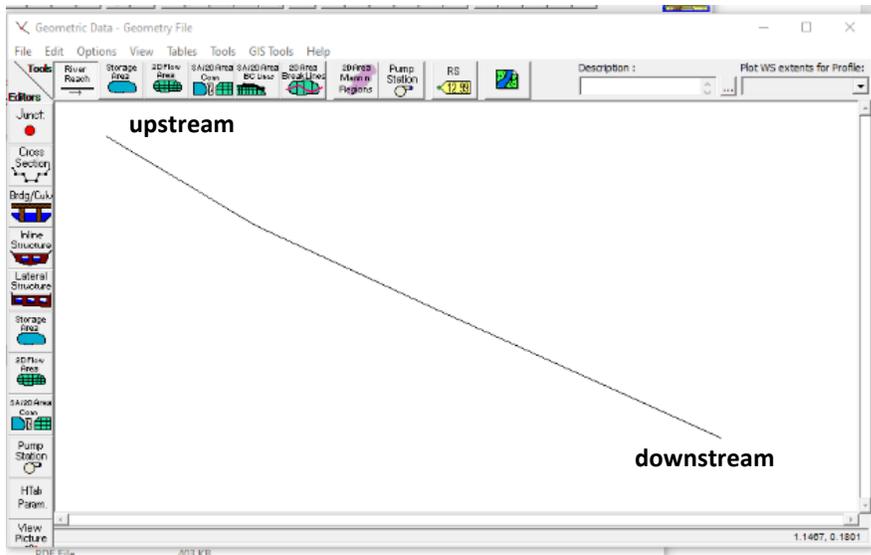
You do not have to provide any name for the geometry file. By default, the geometry file gets the name of the project, and the extension of the file is `.gxx`, where `g` stands for geometry and `xx` is a number. Your first geometry file gets saved as `ras1.g01`. You can give some title (Geometry File) if you want. After you save the geometry file, you notice on the HEC-RAS interface that the geometry box is populated similar to the project box.

Populating Geometry File

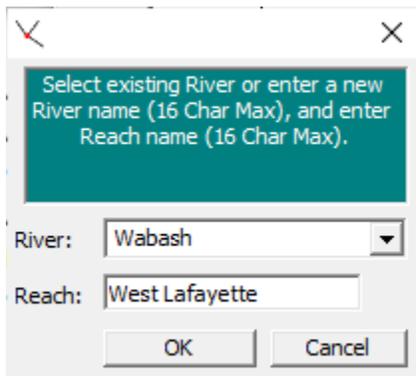
The geometry editor has a file menu (**File**, **Edit**, **Options**, **View**, etc) on top, a list of tools to create River Reach, Storage Area, 2D Flow Area, etc. below the file menu, a list of editing tools to edit junction, cross-sections, bridge/culvert, etc. on the left and the geometry window (blank white area).

Create River/Reach or Centerline

Use the River Reach tool and create a single diagonal reach in the geometry window by starting at the top left corner (upstream end) and ending in the lower right corner (downstream end). Single clicks will create/digitize the reach and double click (at the downstream end) will finish the digitization as shown below.



As soon as you double click, you will be prompted to enter the River and Reach names as shown below. Use Wabash for river name and West Lafayette for reach name. Wabash River runs through West Lafayette, where Purdue University is located, but here we are creating a hypothetical reach!



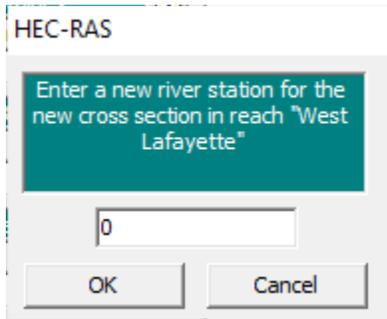
Another very important point to note is that the geometry we are digitizing is not geo-referenced, which means you cannot export the geometry and try to put on a map to match the actual location of Wabash River in West Lafayette. You will learn on how to create a geo-referenced HEC-RAS model later. After you enter the river/reach names, you will see that the reach is created and the flow direction (arrow) is displayed in the geometry window. Note: River/Reach are always digitized from upstream to downstream direction in HEC-RAS.

Save the geometry file.

Create Cross-Sections

Click on the Edit/Create Cross-Section button on the left to get an empty cross-section editor. In the cross-section editor, go to Options→Add a new cross section. The program will ask you to enter a “River Station” for this cross-section. A river station is basically a

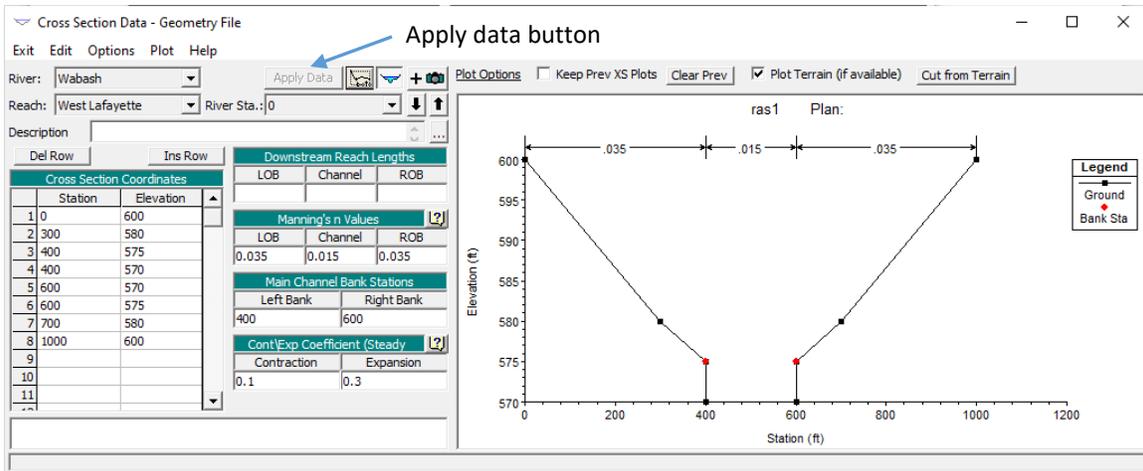
location along the reach. For example, a river station of 50 will mean a point along the river reach at a distance of 50 units from the most downstream point. As our reach is not geo-referenced, a unit of 50 does not mean anything. To make it little more logical, lets use a station of zero (0) so our first cross-section gets created at the most downstream location.



You will see that an empty cross-section at station zero is created. Our next step is to populate the cross-section with the ground profile by providing a series of Station/Elevation values. Use the following values to create a cross-section that is 1000 ft wide by entering the data in the Station/Elevation columns. As our project is created in US Customary units, the values are in ft.

Station (ft)	0	300	400	400	600	600	700	1000
Elevation (ft)	600	580	575	570	570	575	580	600

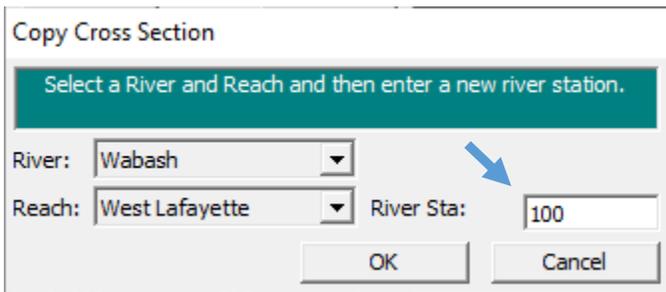
After entering the values, use the “Apply Data” button and you will see a nice looking symmetric cross-section. The cross-section as the main channel of rectangular shape between stations 400-600, and the rest is the floodplain on each side. Next you will see that there are more empty boxes for downstream reach lengths, Manning’s n values and Bank Stations. Since this is our most downstream cross-section (Station = 0), lets leave the downstream reach lengths empty. Use a value of 0.035 for LOB/ROB Manning’s n values and 0.015 for Channel. Use Bank Stations as 400 and 600 for left and right bank, respectively. Use the apply data button and your cross-section should look something like below. You will notice the bank points as red dots at stations 400 and 600, and also the distribution of Manning’s n on top for the main channel and floodplains.



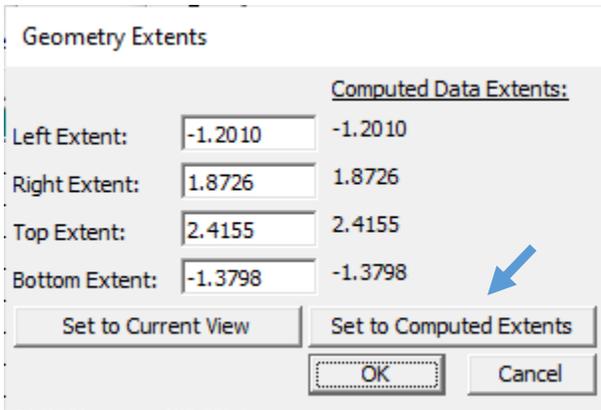
Save your project/geometry data.

If you are happy (I am sure you are!), let's create the most upstream cross-section. As our reach is not geo-referenced, a station of zero will create cross-section at the most downstream location and a station of 100 will create a cross-section at the most upstream location.

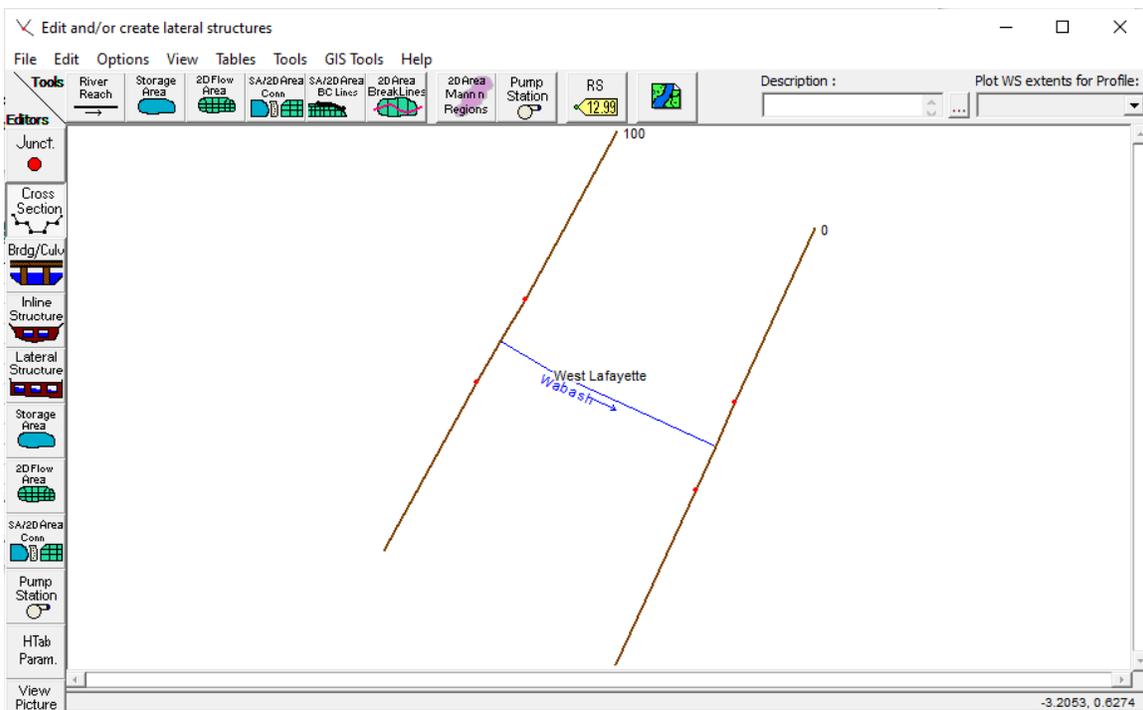
Now that you know how to create a cross-section, you can follow the same procedure to create as many cross-sections as you want using Station/Elevation values. However, to keep things simple, we will replicate the same shape and properties for all cross-sections along the reach. In the cross-section editor, go to **Option** → **Copy Current Cross Section...** All you have to do is enter the station number (100 in this case)



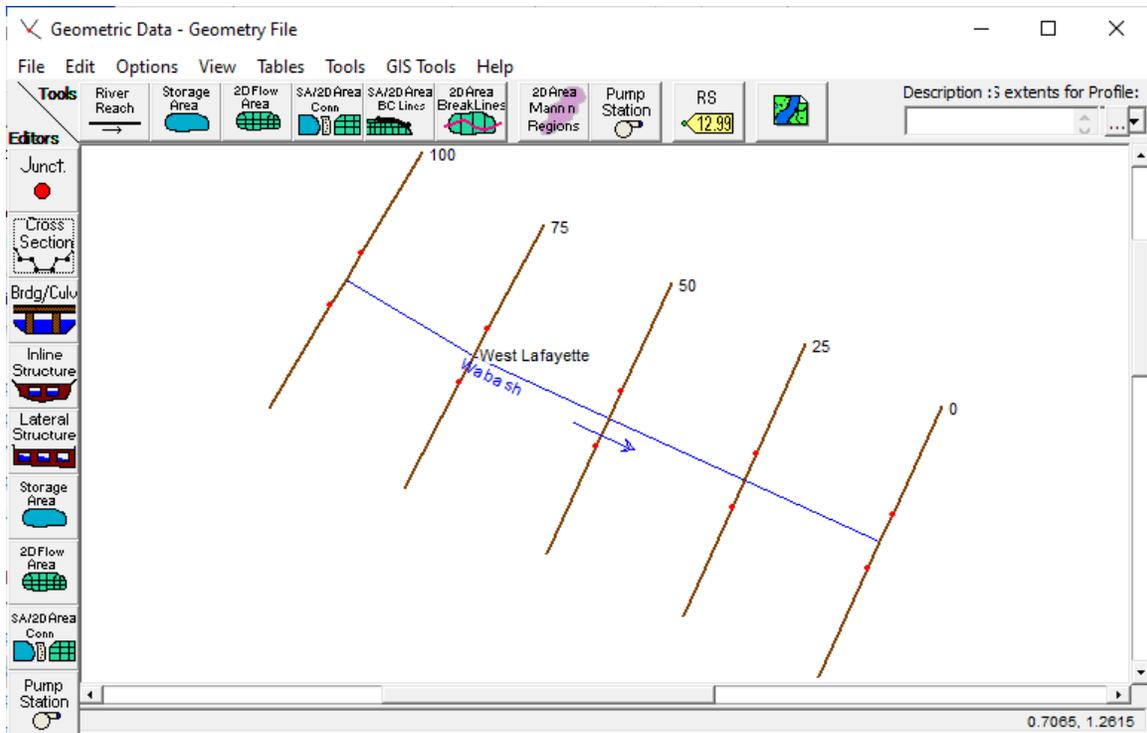
For the new cross-section we will also specify LOB/Channel/ROB downstream reach lengths as 500 ft. Press Apply Data button. After you create/copy the second cross-section at station 100, go back to the geometry editor to see the newly created cross-sections as shown below. If you do not get the complete view of the cross-sections in the geometry window, go to **View** → **Set Schematic Plot Extents..** and change the extent to "Set to Computed Extents" and press OK. You can do this every time you want to set the extent to see the entire geometry.



You should then see your two cross-sections (one at the most u/s end and other at the d/s end). You will see that the stationing along the River/Reach increases from downstream to upstream as the zero station cross-section is located at the downstream end. With respect to cross-sections, the station/elevation values are plotted from left to right looking downstream along the flow direction. The bank points are presented as red dots in the geometry window.



Repeat the same procedure of copying cross-sections to create three more cross-sections at stations: 25, 50 and 75. Exit cross-section editor. You should now see five cross-sections along the reach as shown below.



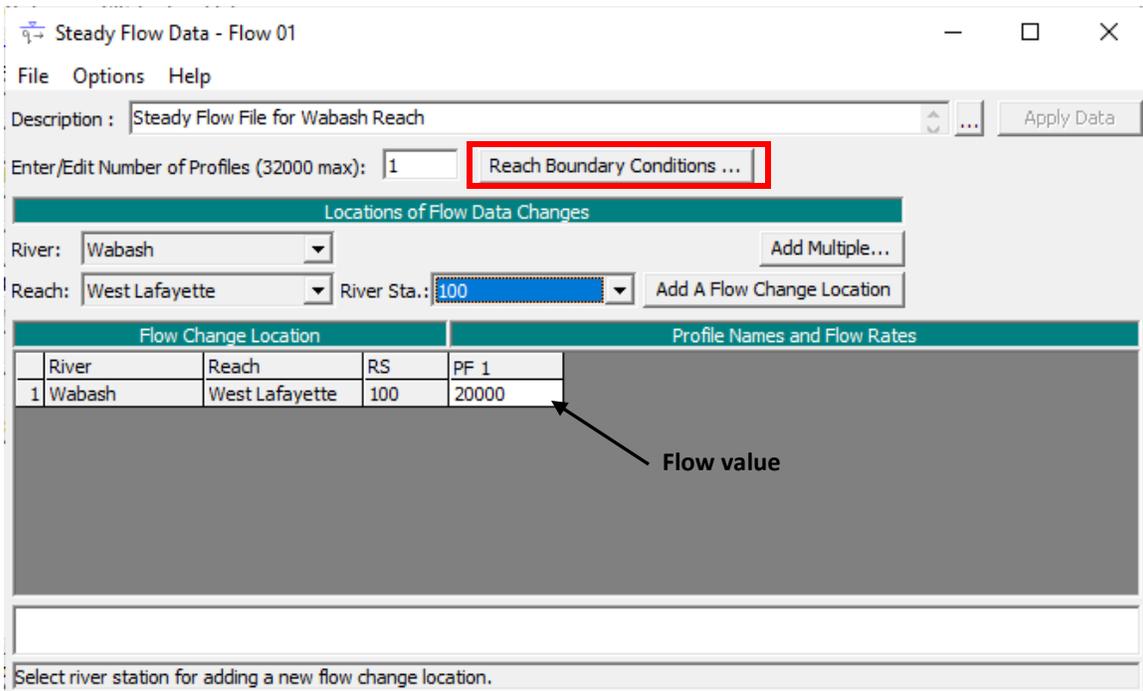
Since our data are not geo-referenced, the stationing along the reach is only proportionately applied here, but the stationing along the cross-sections is in feet as we need this information in proper units to compute cross-sectional areas and flow conveyance. Also the distance specified in the LOB/Channel/ROB is in feet for computing the loss term in the energy equation. Thus, technically our reach is 2000 feet long. Close the cross-section editor, save the geometry file and exit the geometry editor.

Now that the geometry is ready, we can specify a discharge value and let it flow through the reach to get the water depth or elevation and velocity at each cross-section.

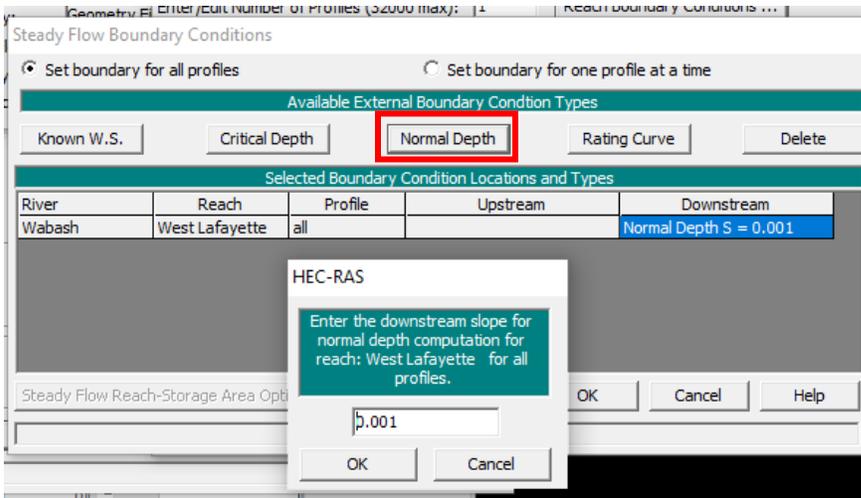
Creating Steady Flow File

Discharge value(s) in HEC-RAS are specified using a Steady/Unsteady Flow file. In this lab, we will create a steady state simulation. Steady state means the flow is not changing with time so we will use only one value. Typically, for most design purposes, when we have the discharge hydrograph, we will use the peak value to simulate the river hydraulics in HEC-RAS. To create a steady flow file, in the main HEC-RAS interface, go to Edit→Steady Flow Data..

You can enter some description (Steady Flow File for Wabash Reach). We will supply an upstream boundary condition of flow or discharge at Station 100 as 20,000 cfs. Press “Apply Data” button. PF 1 stands for profile flow 1. You can use the “Enter/Edit Number of Profiles” box to enter the number of profiles or flow values, but in the lab we will only create one flow profile.



Next you will use the Reach Boundary Conditions.. (highlighted in the figure above) button to specify the downstream boundary condition at station zero. We will use a simple “Normal Depth” boundary condition and specify a bed slope of 0.001 as shown below.

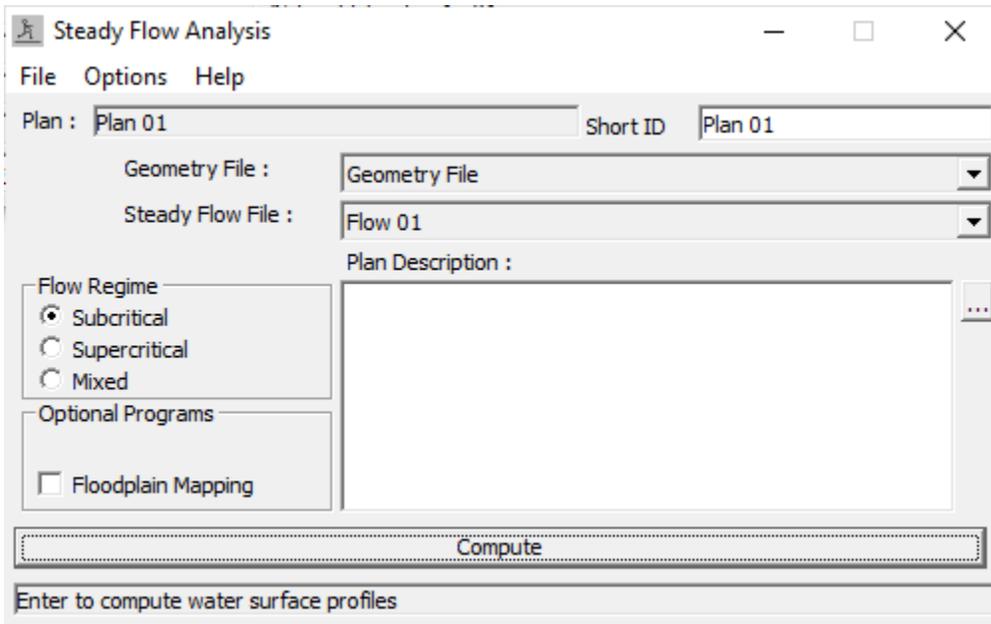


Press OK for both boundary condition window and save the steady flow data by going to file save flow data. In the main HEC-RAS interface, you will now see that ras1.f01 (f for flow and 01). Once the flow data are complete, the model is ready for simulation!

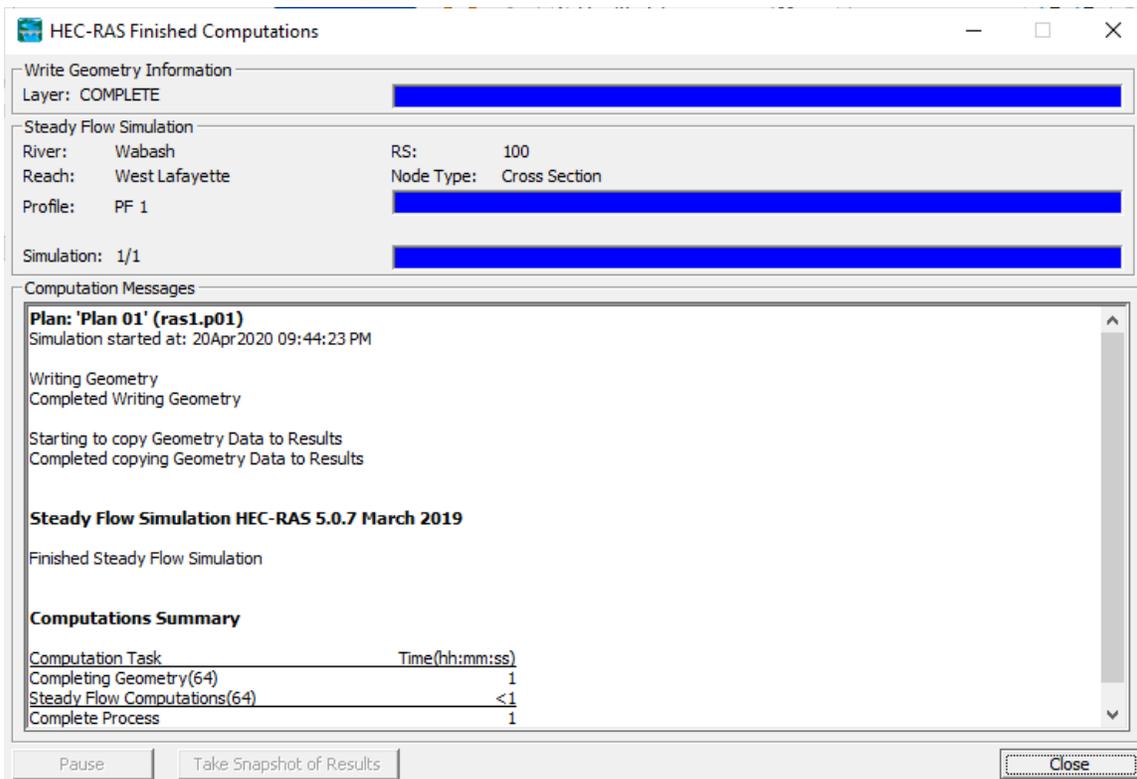
Running Steady State Simulation

To create a steady flow simulation, on the main HEC-RAS interface, go to Run→Steady Flow Analysis. As soon as you do this, a plan file also gets created and it stores the

geometry and flow information used in running the simulation. The main HEC-RAS interface will show this plan file. In the Steady Flow Analysis window, change the flow regime to “Subcritical”, and press the Compute button.



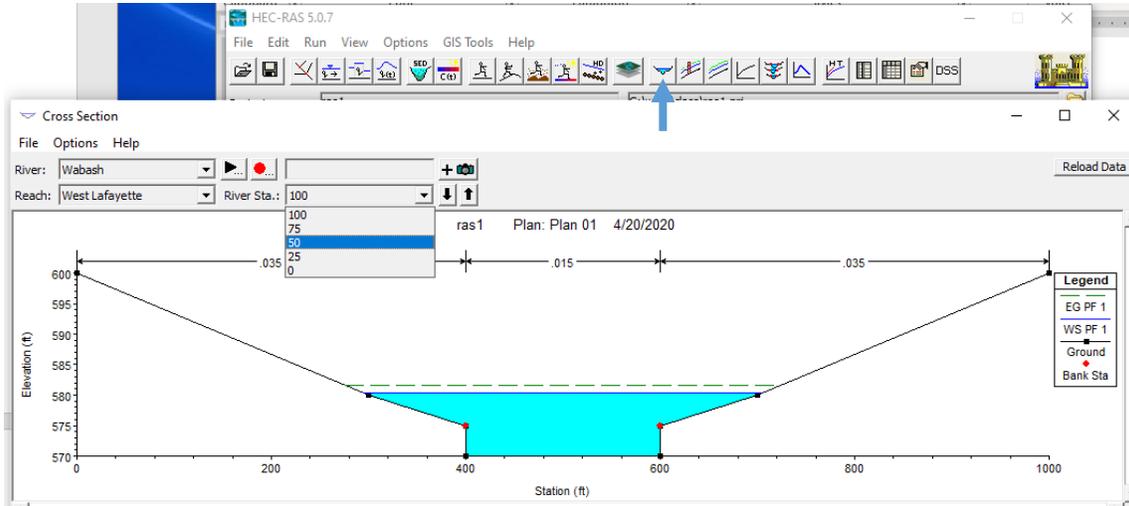
It should take only a few seconds to complete the simulation.



We can now visualize the results.

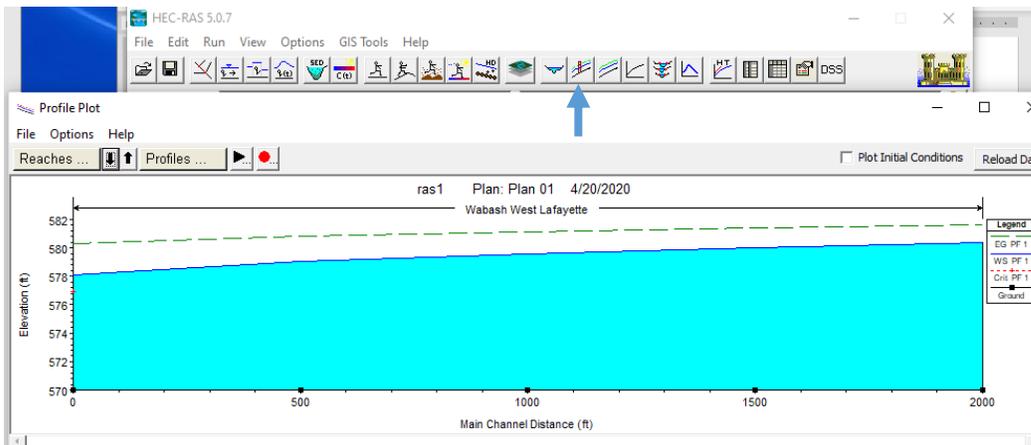
Visualizing HEC-RAS Results

Cross-sectional View: This can be done using the cross-section view button (shown by arrow below) on the main HEC-RAS interface.

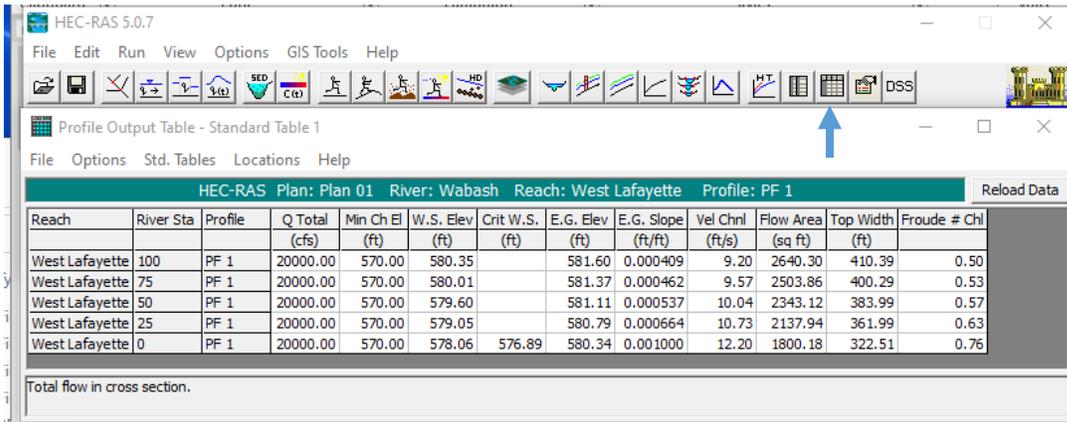


In the cross-sections view window, you can change the station number to see the water level at different cross-section.

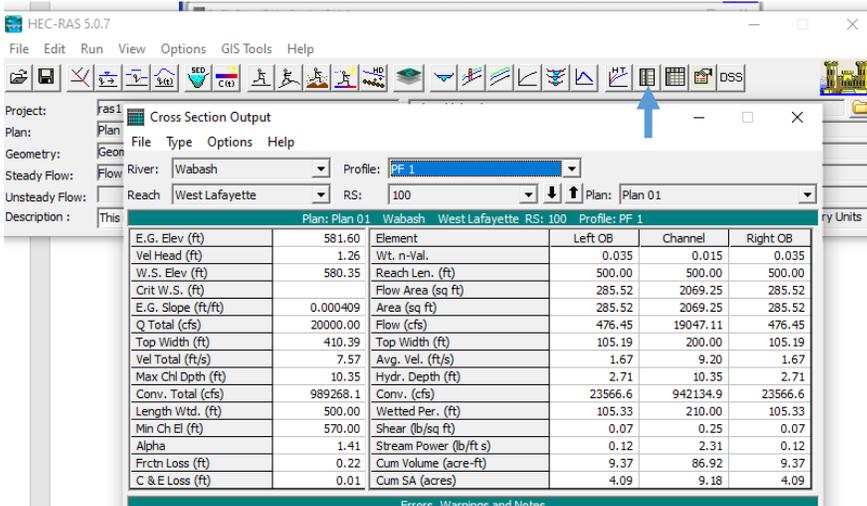
Profile View: This can be done using the profile view button (shown by arrow below) on the main HEC-RAS interface.



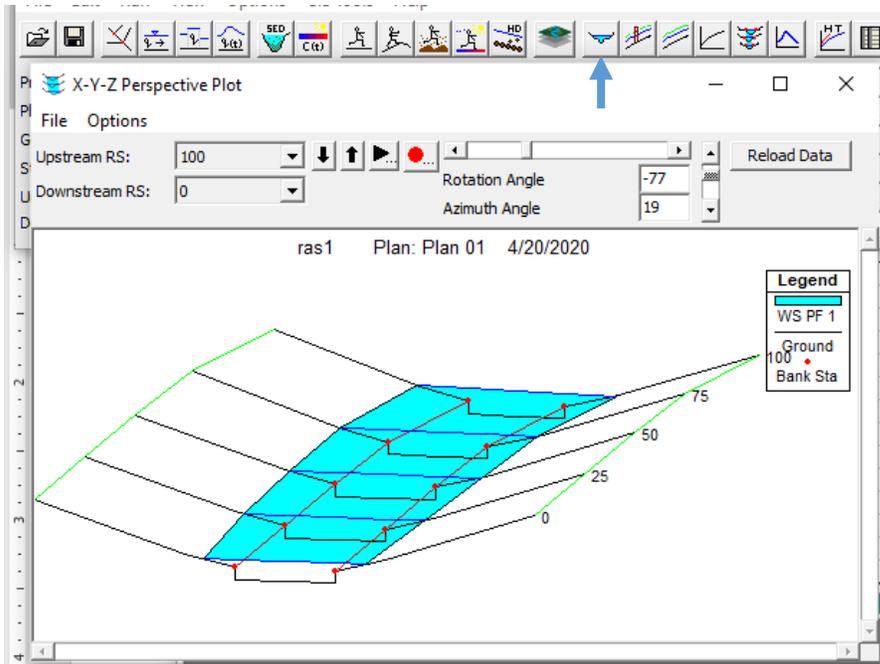
Tabular View: There are two options. The first option is to view the Summary Output Table for the entire profile by using the summary table output button (shown by arrow below). This gives an overview of channel average properties for each cross-section in one table.



The second tabular view is the detailed output at each cross-section by using the detailed cross-section output button (shown by arrow below).



You can also view a xyz (3D) profile plot by using the 3D output button as shown below:



Save your project. Okay, you are done with your first Introduction to HEC-RAS tutorial! Congratulations!!

Homework

1. Report the average water surface elevation for the entire profile (reach) in ft for your simulation.
2. Keeping the Manning's n for floodplain at 0.035, change the channel Manning's n to 0.03 for all cross-sections and report the average water surface elevation for the profile in ft.
3. Keeping the channel Manning's n for channel to 0.015, change the floodplain manning's n for all cross-sections to 0.06 and report the average water surface elevation for the profile in ft.
4. Discuss how the change in Manning's n for the channel and floodplain affected the water surface elevation. Was the water surface elevation more sensitive to channel manning's n or floodplain manning's n ? why?
5. Repeat 2 and 3 above with a steady flow of 100,000 cfs. Discuss how the change in Manning's n for the channel and floodplain affected the water surface elevation. Was the water surface elevation more sensitive to channel manning's n or floodplain manning's n ? why?

(Note: Look at the video to see how Manning's n can be changed for all cross-sections at once)