HEC-RAS in 5 steps

Hec-Ras has been developed by the Hydrologic Engineering Center for the U.S. Army Corps of Engineers. This is a brief summary of the 5 main steps to run the model for the simplest case. This guide has been elaborated to be used together with the Flood exercise "2. Learning Hec-Ras".

1. Downloading the software:

http://www.hec.usace.army.mil/software/hec-ras/

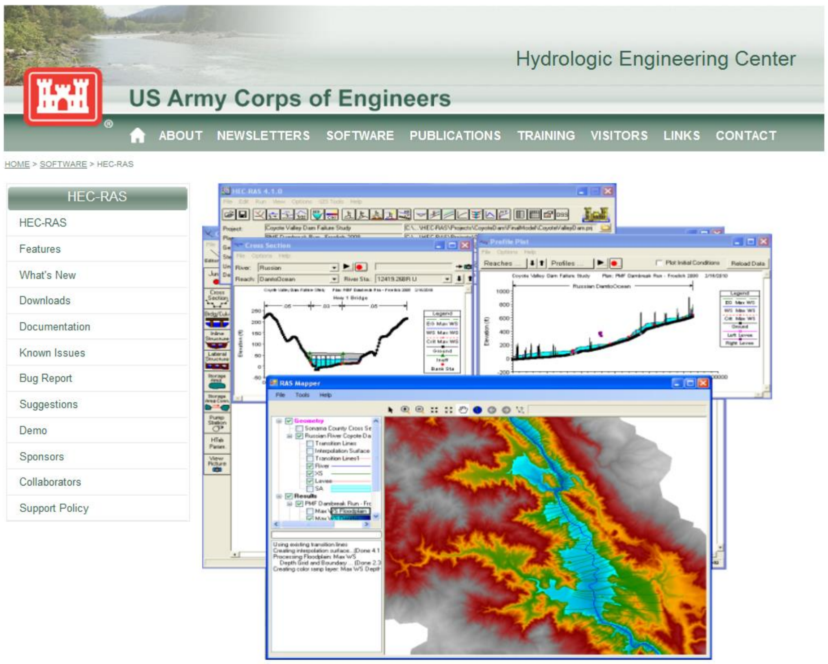

Figure 1: Hec-Ras webpage

Download (Fig. 1) and install the software.

You can download the executable code, documentation (User's Manual, Hydraulic Reference Manual) and **examples** (explained in the Application Guide). It is highly recommendable to do all the examples to learn different aspects of Hec-Ras. In this brief document we are just going to learn the first steps for the simplest case, an idealized single river reach. To make a one-dimensional hydraulic simulation it is necessary to have a solid base in hydraulics, river dynamics and numerical modelling.

2. Starting Hec-Ras:

To run the model we need to follow some steps, the first one will be to establish the working directory and create a new project (Figs. 2 and 3).

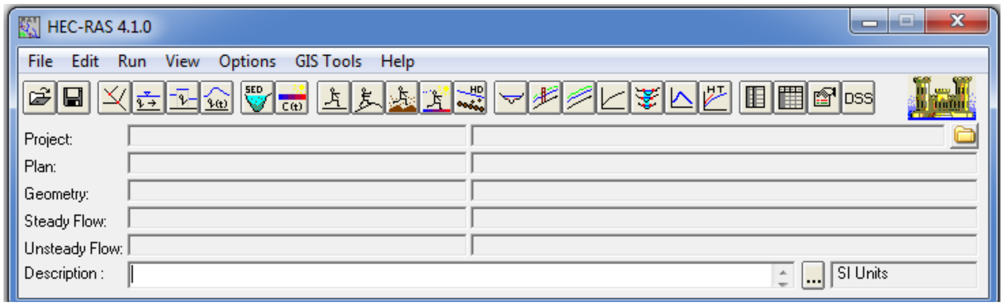

Figure 2: Main window Hec-Ras 4.1.0

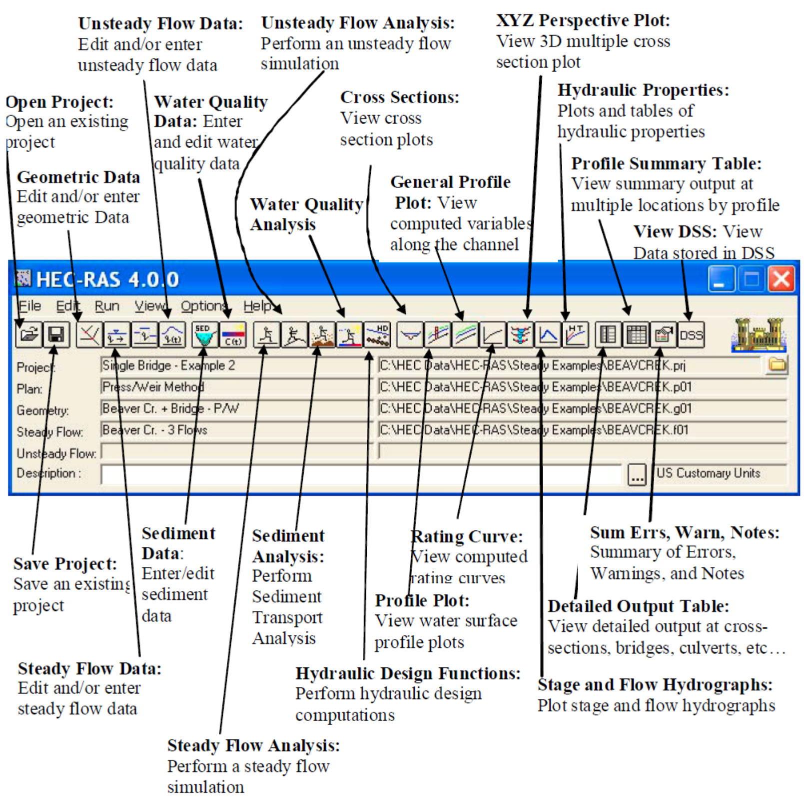

Figure 3: Main window button bar (from Hec-Ras User's Manual)

Saving the new project: **File_New Project**: Select the Title of the project and the folder to save it.

We need to check the Units System**: Options_Units System** (<u>should be SI metric</u>)

3. Creating a hydraulic model:

We can define the Geometric Data, which is the stream system, cross sections data and infrastructures (i.e. bridges).

Here the model set up will be based on the exercises. The model has much more options that should be explored with the help of the Hec-Ras examples. In addition, the tool Hec-GeoRas (see website) can be used for processing geospatial data in ArcGIS using a graphical user interface (GUI). The interface allows the preparation of geometric data for import into Hec-Ras and processes simulation results exported from Hec-Ras.

To create the river geometry we need to draw the river reach (River Reach icon), give the name (to the River and the studied reach), and define the cross sections (Cross Section icon; Fig.4).

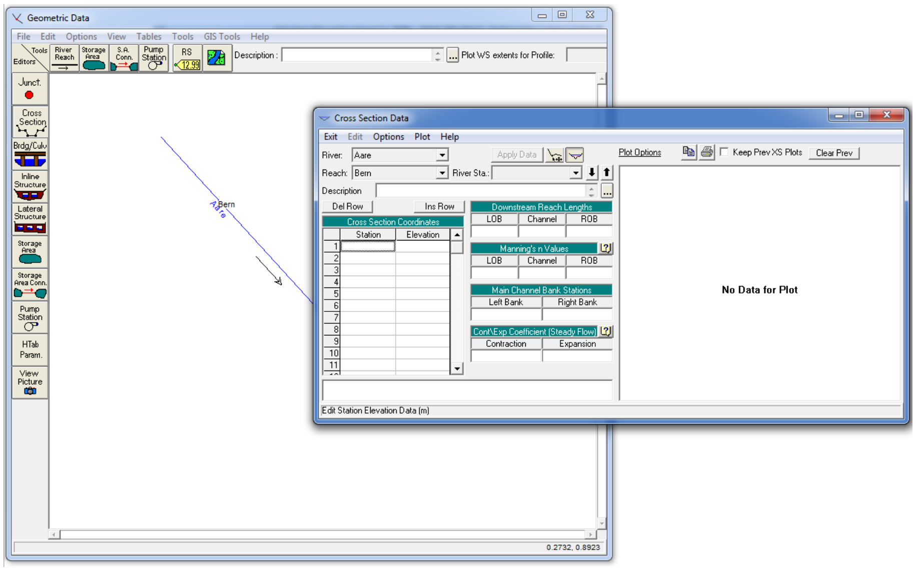

Figure 4: Geometric Data and Cross Section windows

To inter the cross sections data we need to add the profiles (elevation and distance from the left bank) in the Cross Section Coordinates table.

Then, we need to define the Downstream Reach Lengths for both left and right bank and the main channel. This will be 0 m if we are defining the most downstream cross section (there is not any other section downstream of this one).

The bank statins need to be defined (distance from the left margin). If they are not already defined in the coordinates table the program will ask us if we want to interpolate them.

Finally we need to define Manning roughness values for each bank and the main channel (Fig.5). There is a helpful table in the Manual if we click in the question mark 

. There are many guides with reference values (USGS among others), and bibliography to check (Chow, 1959; Barnes, 1967). We can calculate the roughness values using empirical equations (such as Stickler, 1923; Limerinos, 1970). As it was explained in the Theory lecture, this is one of the most important parameters in river hydraulics, and **should be calibrated if possible**.

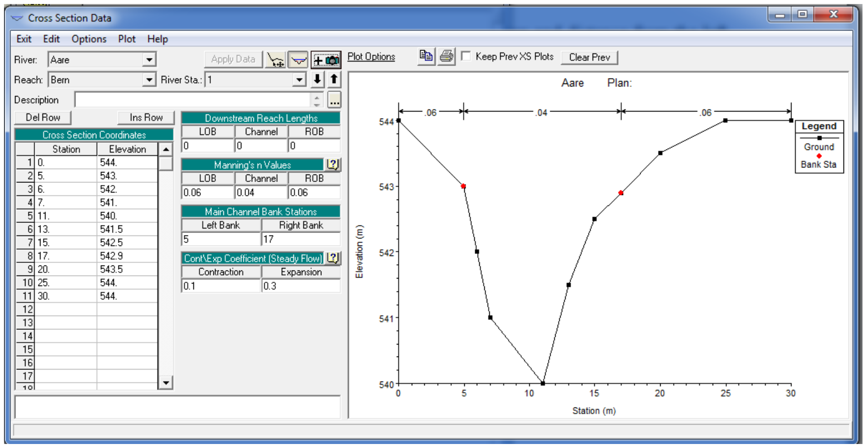

Figure 5: Cross Section Data Window

For the proposed exercise we will copy the data from Cross Section 1 to Cross Section 2 (**Options_Copy Current Cross Section**), changing only the elevation (**Options_Adjust Elevation**) and the Downstream Reach Lengths.

To visualize the cross section geometry we click Apply Data, and to visualize the reach geometry we exit from Cross Section Data window. Then we save the geometry (Fig. 6).

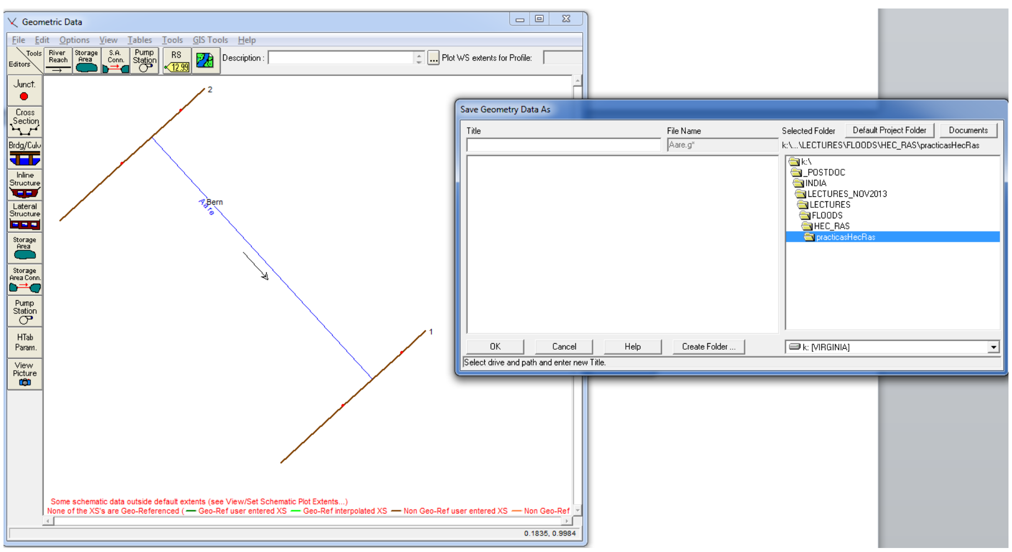

Figure 6: Geometric data window and saving the Geometry Data

When the geometry is finished we need to enter the Flow Data and Boundary Conditions. In our exercise we compute steady flow, so we will select the **Steady Flow Data Editor** (Fig.7) from the main window.

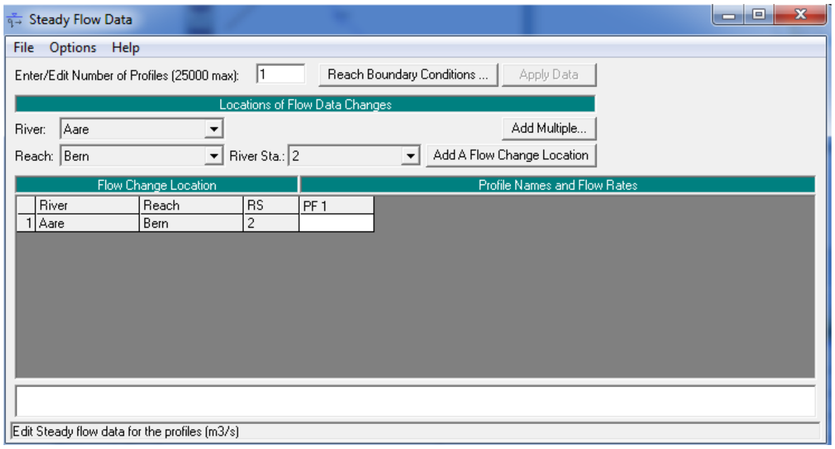

Figure 7: Steady Flow Data window

We can compute several numbers of Profiles (hypotheses or scenarios). Flow values must be entered for all profiles. At least one flow must be entered for every reach, but this can be changed at any location within the river system.

Boundary conditions are required. As explained in Theory, if subcritical flow is performed (as in our case), only the downstream boundary condition (critical depth) is needed (Fig.8). The proper setting of boundary conditions is very important, since results will be modified according to them. It is required a good knowledge of the study reach and the flow conditions we want to simulate.

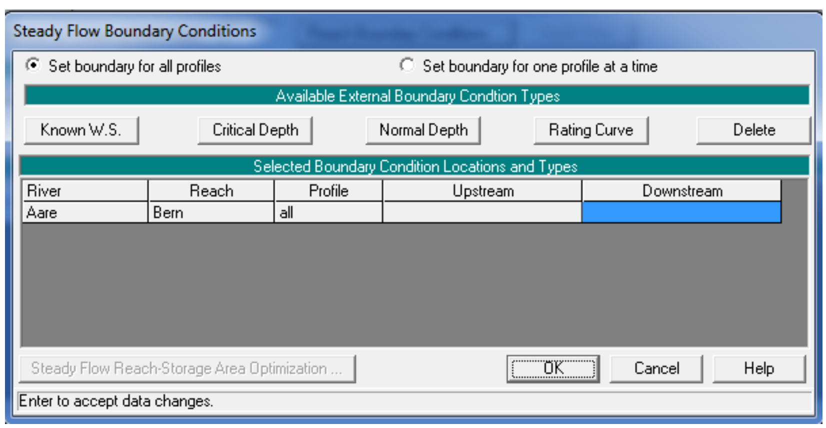

Figure 8: Steady Flow Boundary Conditions window

4. Performing the hydraulic computations

Once we established the boundary conditions we save the flow data and we can compute the model clicking **Run_Steady Flow Analysis** in the main window (Fig.9).

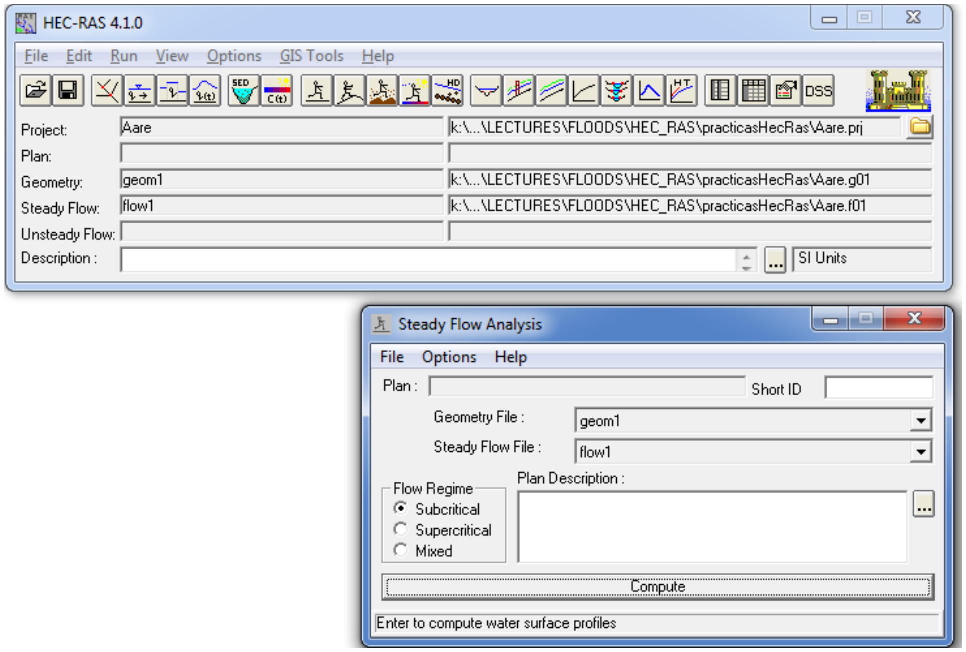

Figure 9: Steady Flow Analysis window

There are additional features (Options menu) out of the scope of this exercise, so students are encouraged to revise Hec-Ras manuals and examples.

5. Analysing the results

Several outputs features can be analysed. These options include: View Cross Sections, Water Surface Profiles, Perspective plot, rating curve, etc (Fig.10).

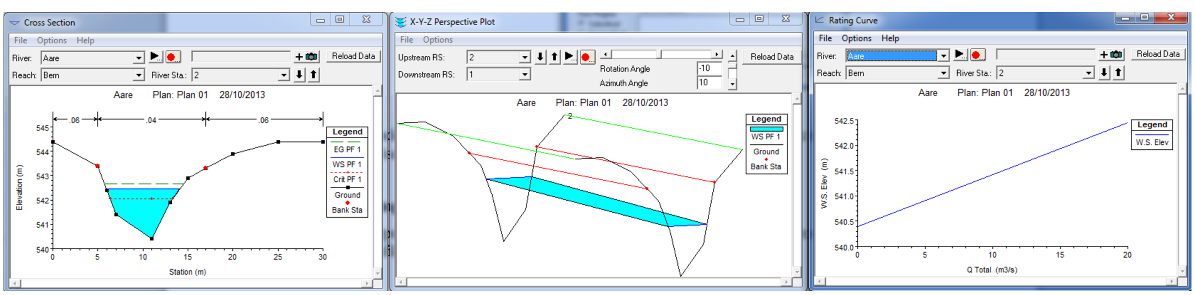

Figure 10: Result windows