HEC-RAS

A Tutorial

(Model Development of a Small Flume)

HEC-RAS

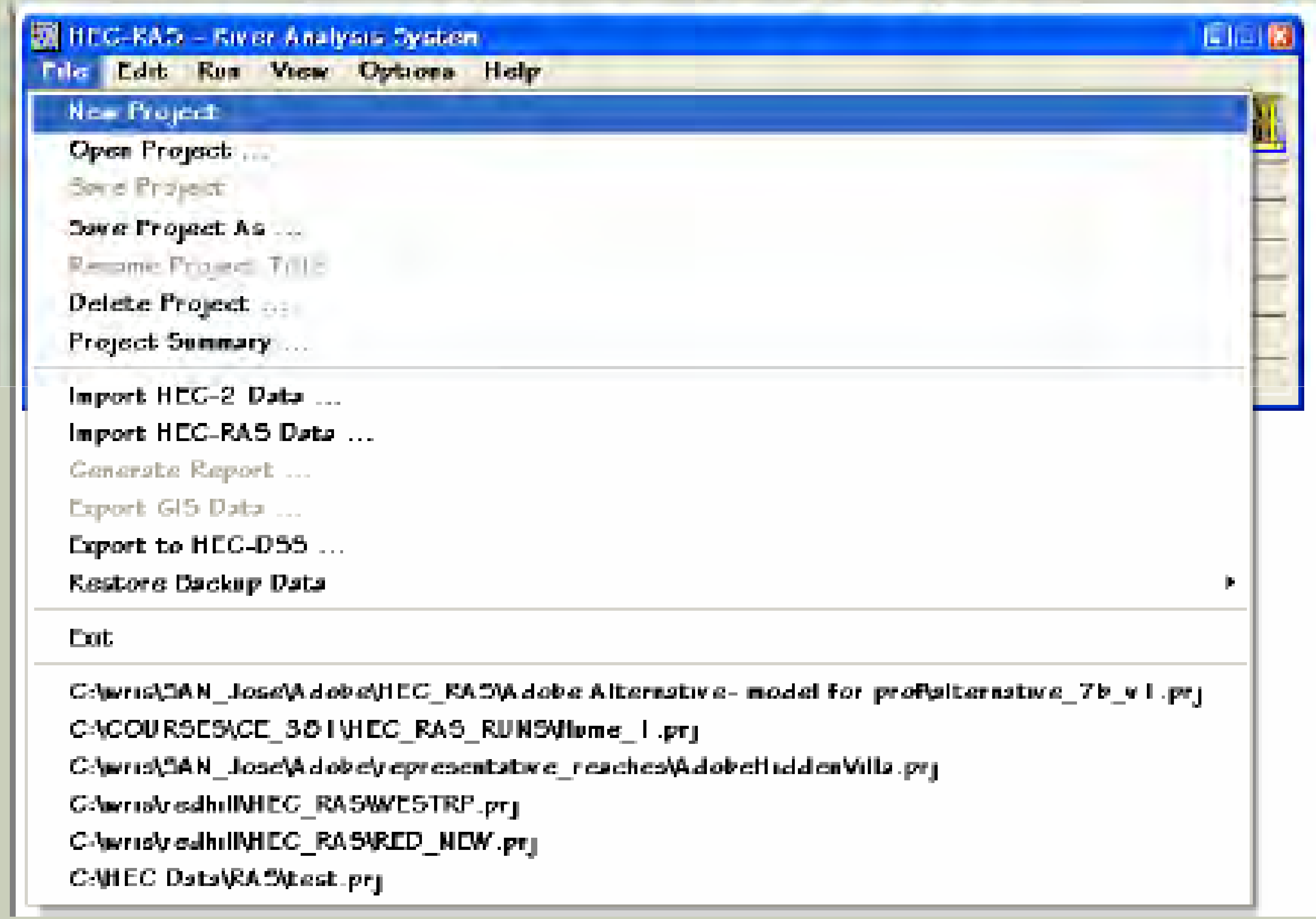
- Hydraulic Engineering Center:River Analysis System
- 1-D step backwater model
- Utilizes energy equation to compute water surface elevation for a given discharge, geometry, and resistance

Organizational Structure

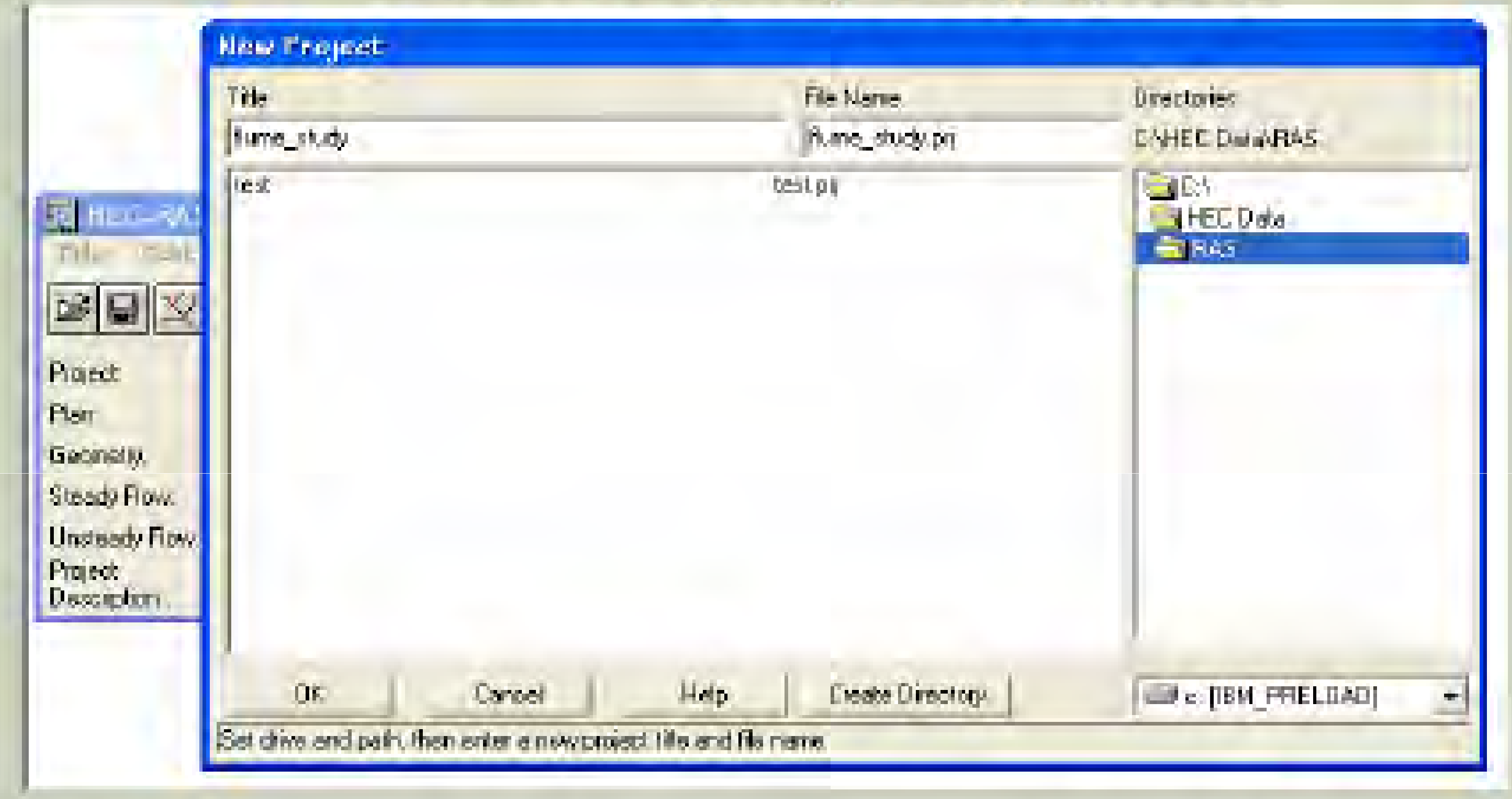
- Open project –creates (.prj file)
- Includes:

1. Unit system (SI/US customary)
2. Geometry (XS, bridges, weirs, etc.) (.g file)
3. Flow Data (steady, unsteady) (.f file)
4. Plan data(what combination of flow/geometry to use for the analysis) (.p file)

Open a new project

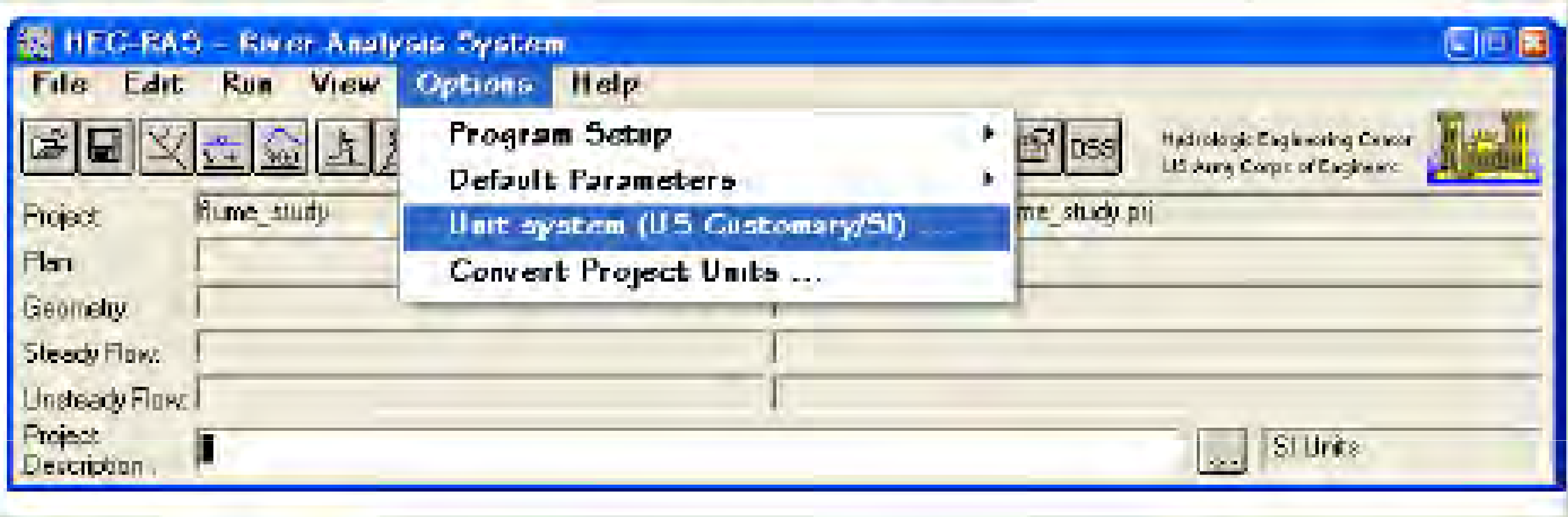


Name the project (.prj file)



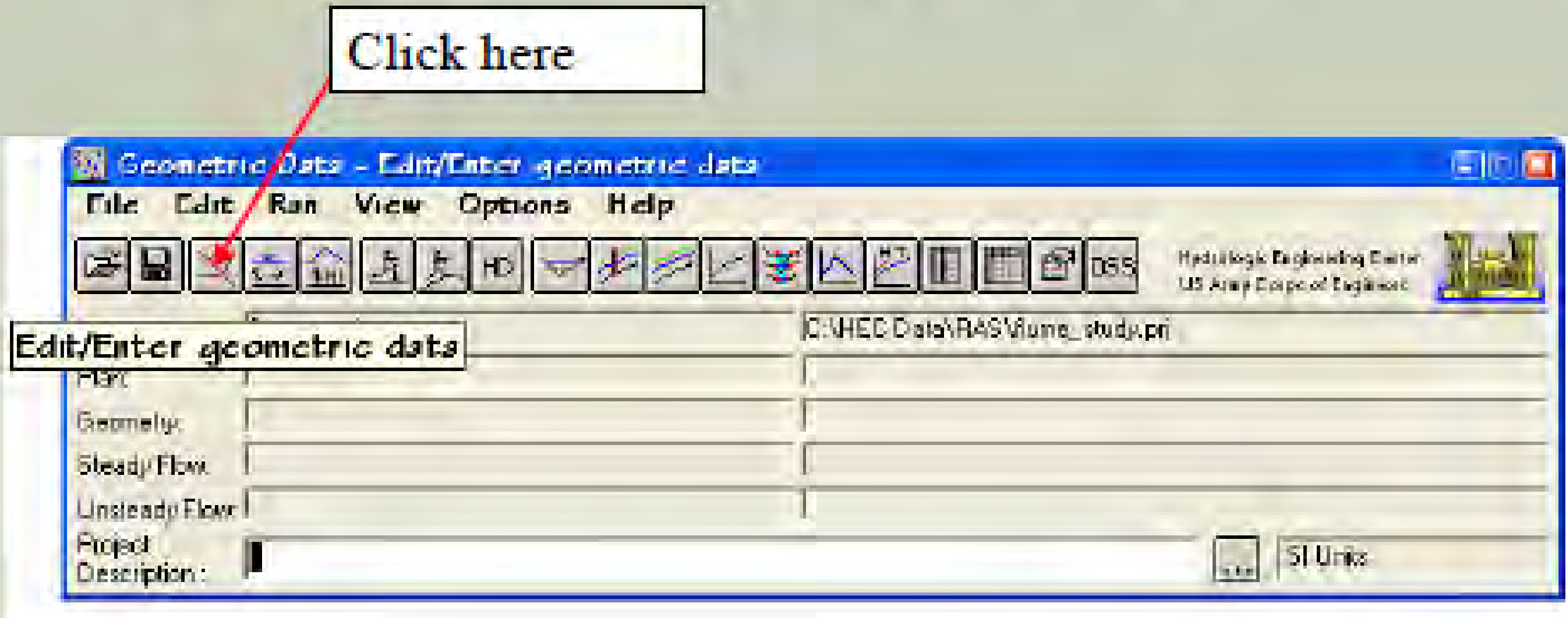
After you have entered the above, click O.K. and O.K. again

Select SI units

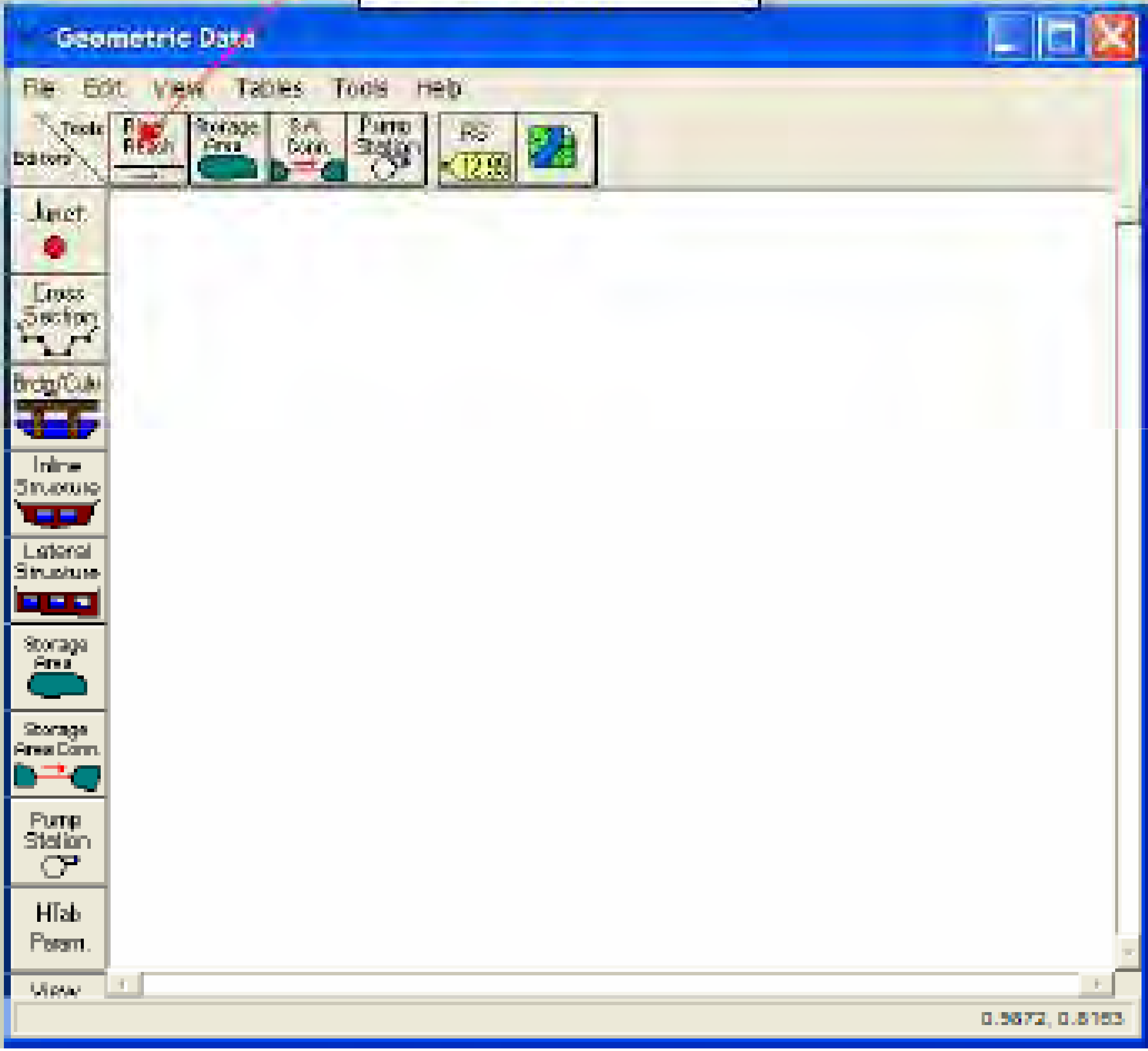


Geometry Files (.g)

- Create a reach- single or dendritic

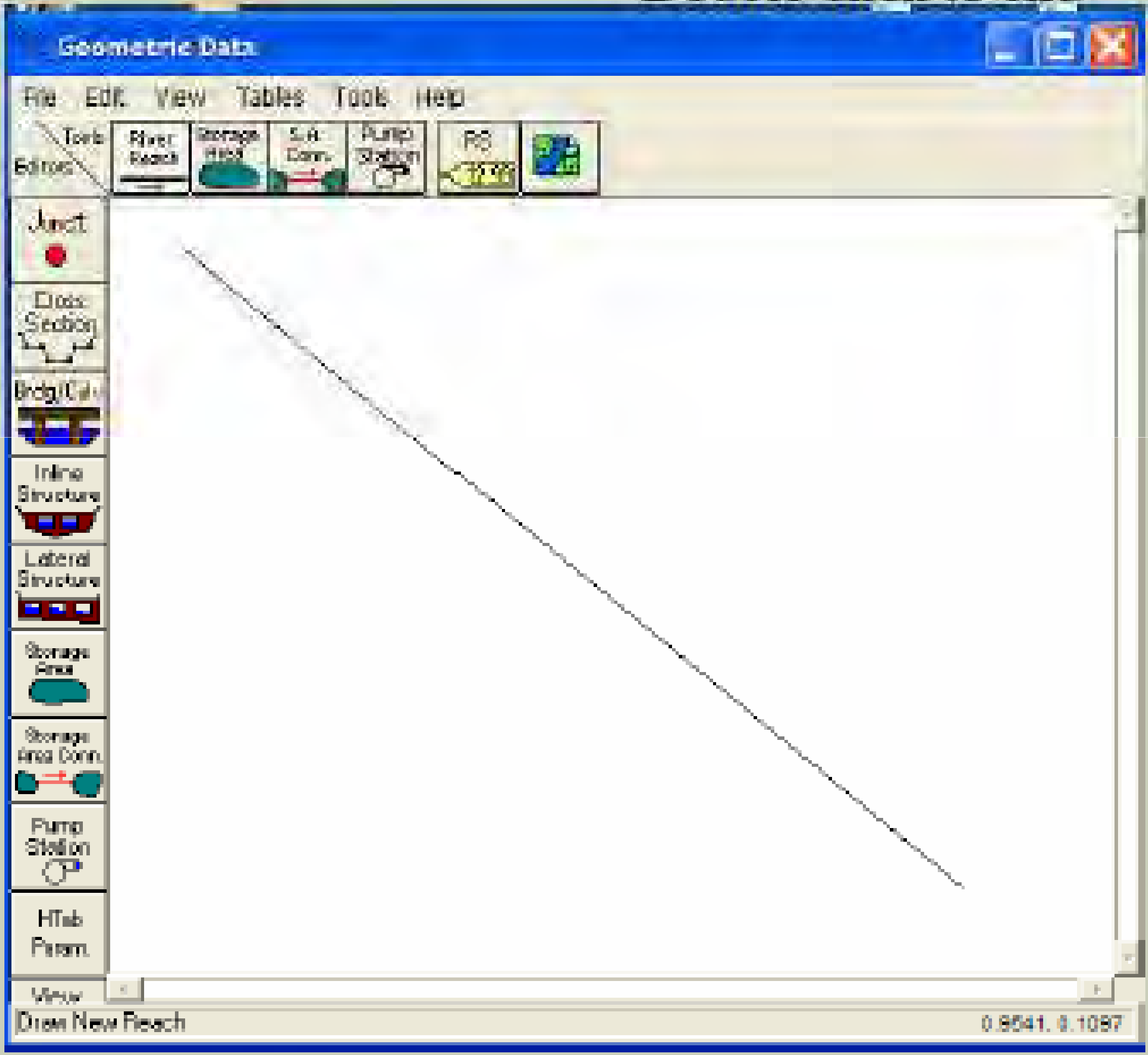


Create a new river reach



Draw the reach upstream to downstream

Double click to end



Name the river and reach

Geometry

- Cross sections define the channel/flume geometry
- Cross sections are defined by Station(x) and elevation (y) in the plane of the cross section perpendicular to the flow
- River station, downstream reach length and cross section thalweg elevation define the channel slope
- Overbank stations differentiate channel and floodplain characteristics
- Resistance to flow is defined by Manning n coefficients for both the channel and floodplain
- Expansion and contraction coefficients define energy losses associated with velocity head changes between cross sections
- Ineffective flow areas are can store but not convey water downstream
- Obstruction areas block flow completely
- Levee elevations confine flow to channel until the levees are overtopped

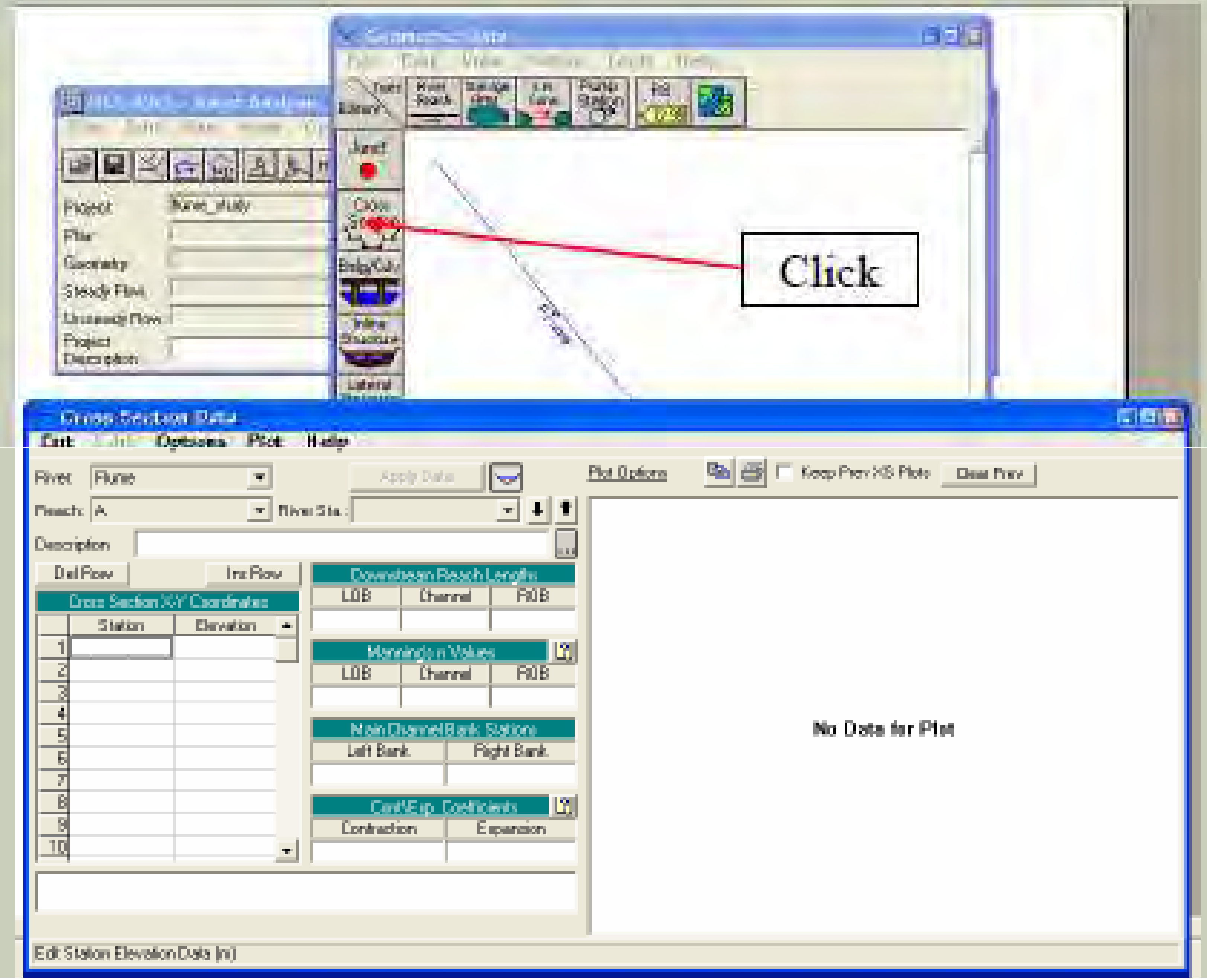
More geometry

- Junctions define where two reaches are connected
- Bridges and culverts
- Inline weirs/gates
- Off channel storage
- Pumping stations
- All can be modeled with a reasonable amount of detail (can be covered at a later date)

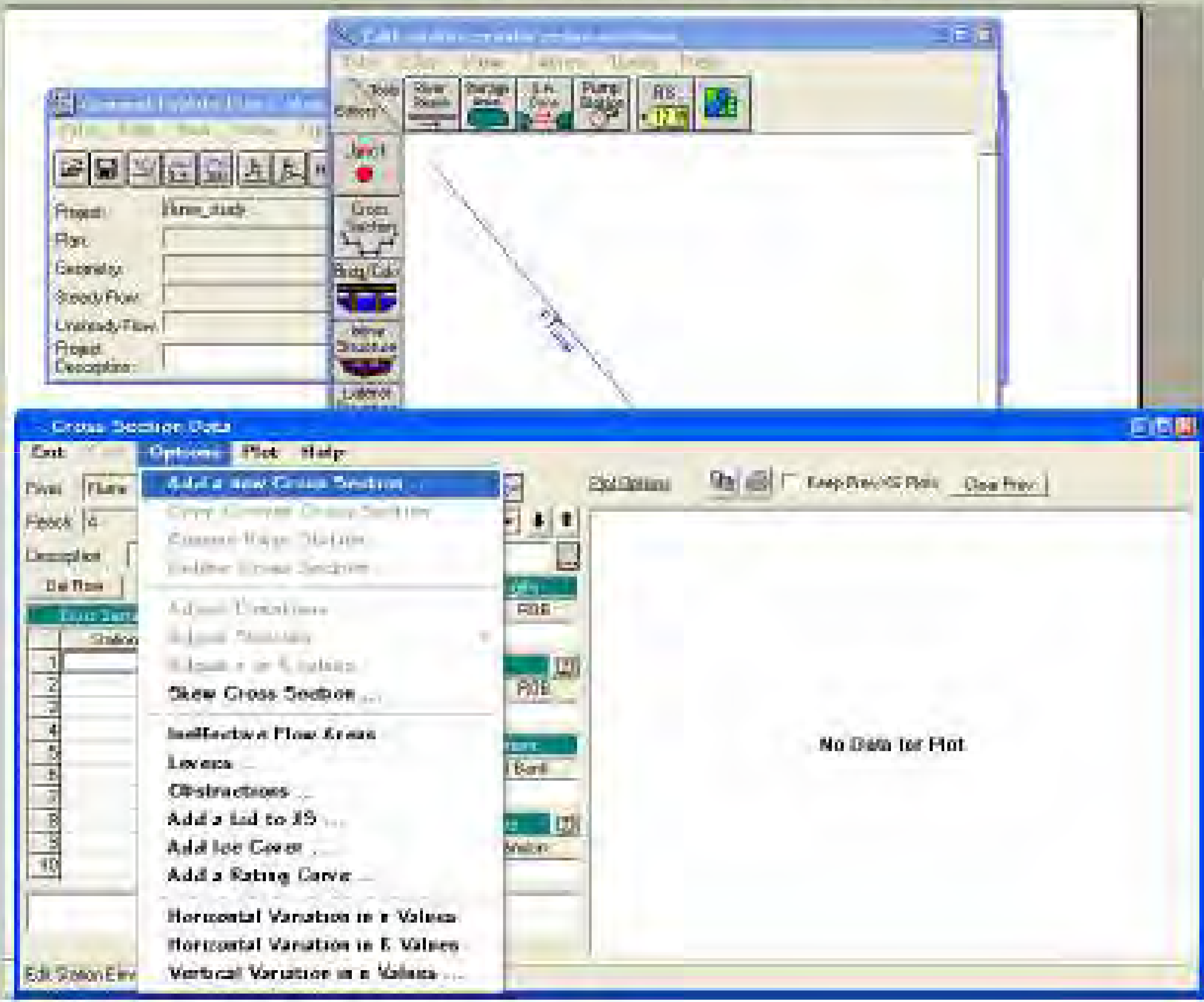
Flume example

- Width- 7.57 cm (0.0757 m)
- Wall height- 12 cm(0.12 m)
- Flume Length- 182 cm (1.82 m)
- Slope-none , horizontal
- Roughness- Plexiglass (n~0.0085)
- Discharge- 1.5 l/s (0.0015 m^3/s)
- Cross section spacing –10 cm (0.1 m)
- Downstream boundary condition (critical depth at outlet, free overfall)

Enter cross section data

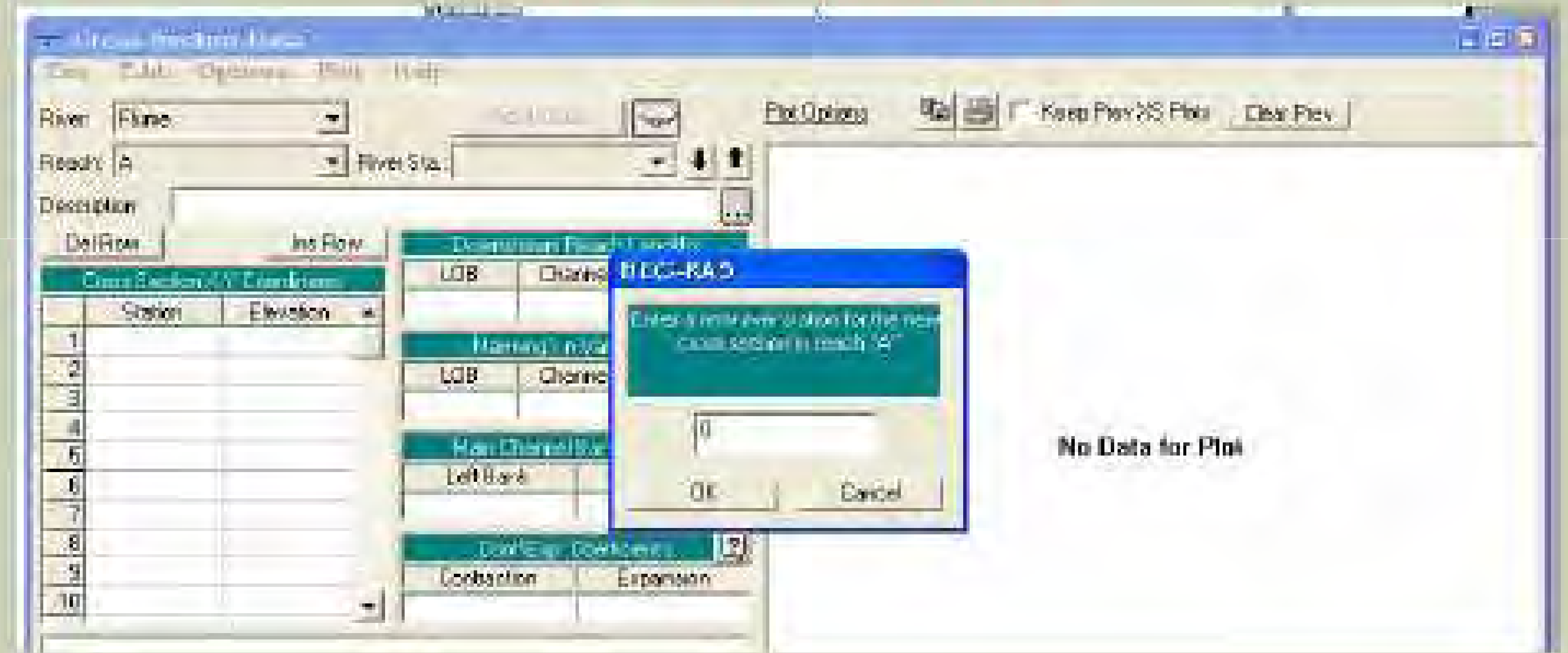


Add a new cross section



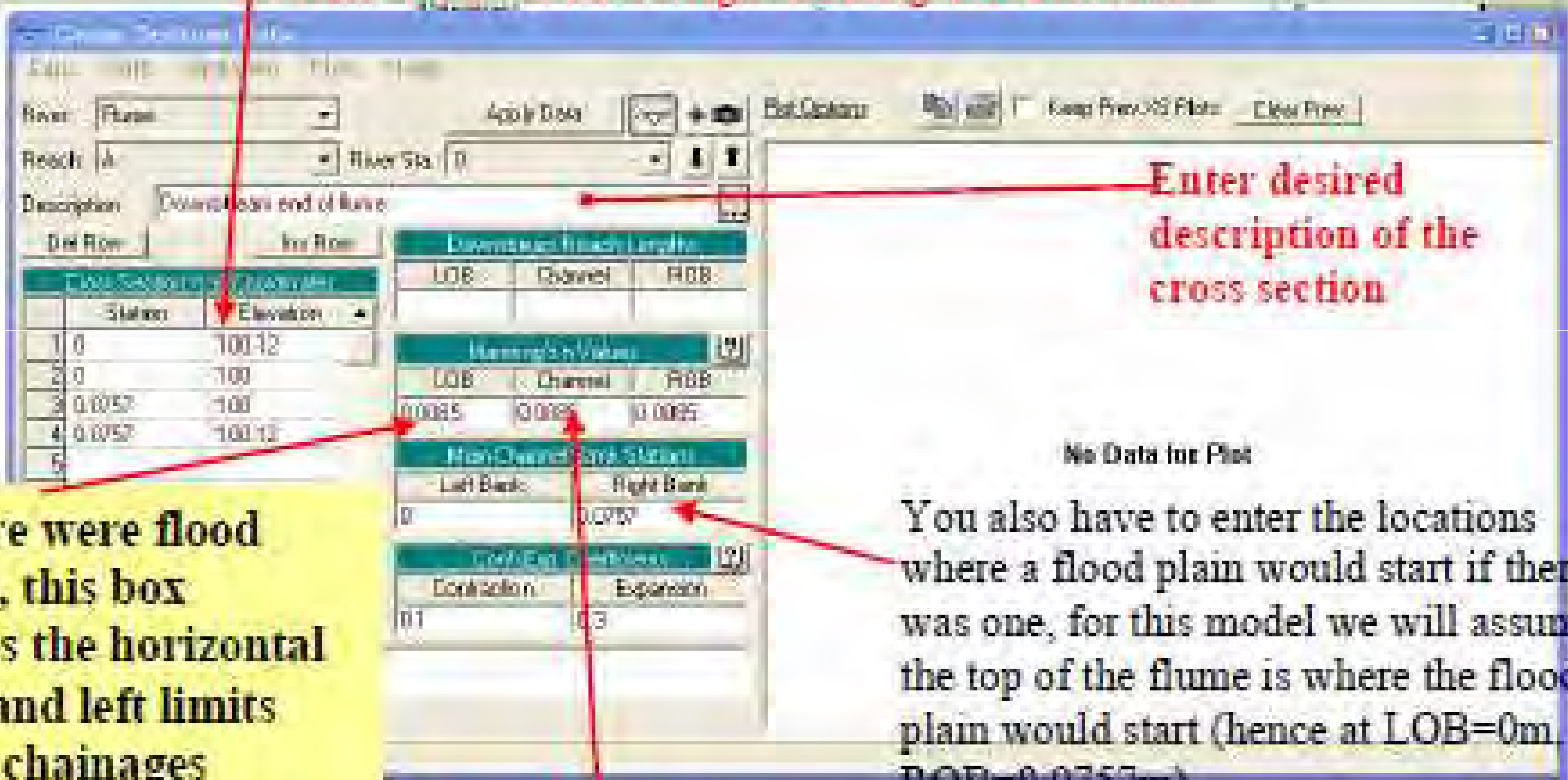
Start at downstream river station 0.00 m

(HEC RAS ESTABLISHES THE DOWN STREAM END AS ZERO FOR A STARTING DISTANCE AN PROGRESSIVELY WORKS UP STREAM IN CHAINAGE)



Enter and apply data assuming an arbitrary datum of 100.00 m, notice the downstream reach length is zero since we are at the furthest downstream extent of the model

Cross-sectional data
- Entered from left to right looking down stream



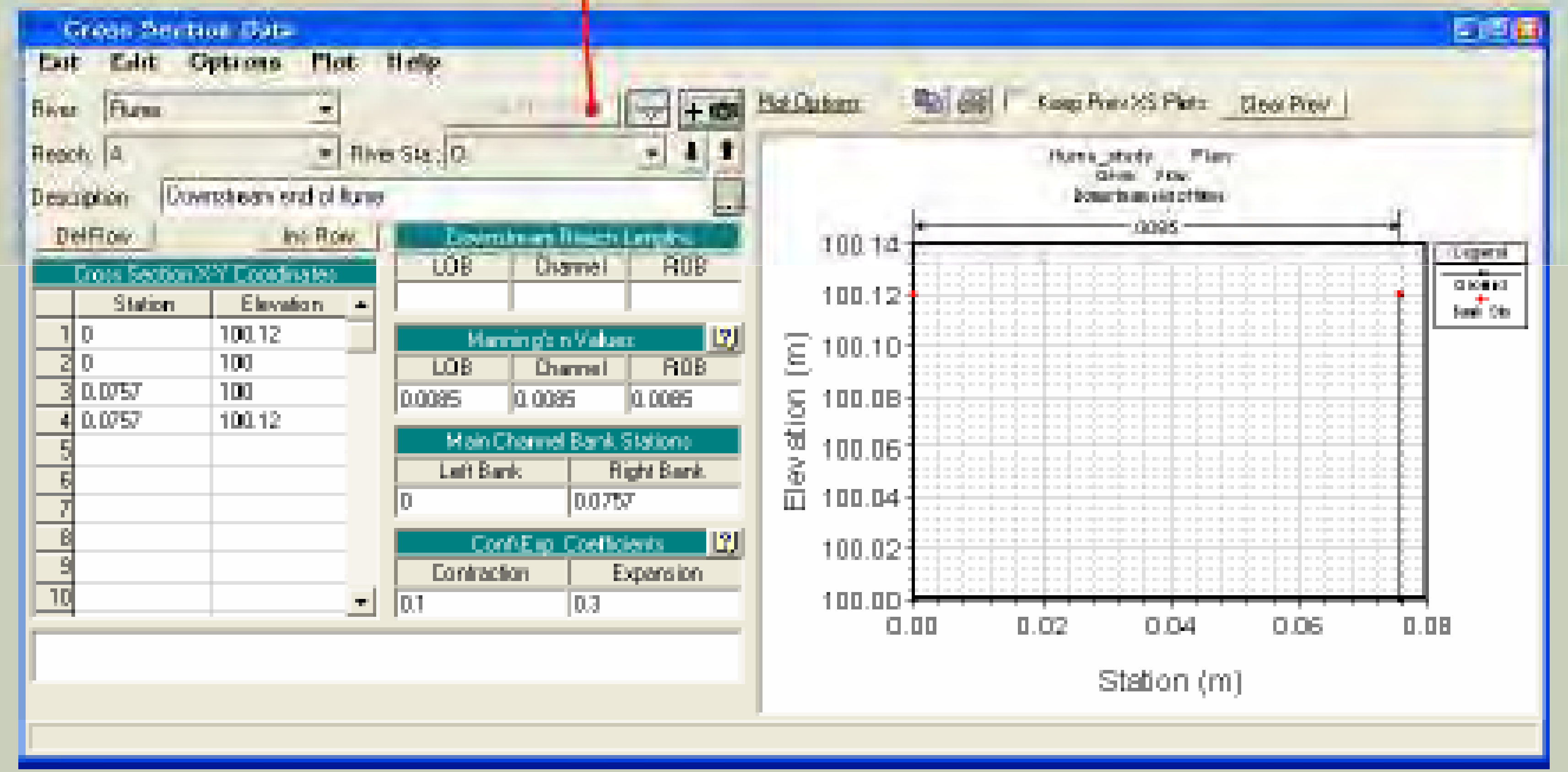
Enter desired description of the cross section

You also have to enter the locations where a flood plain would start if there was one, for this model we will assume the top of the flume is where the flood plain would start (hence at LOB=0m, ROB=0.0757m).

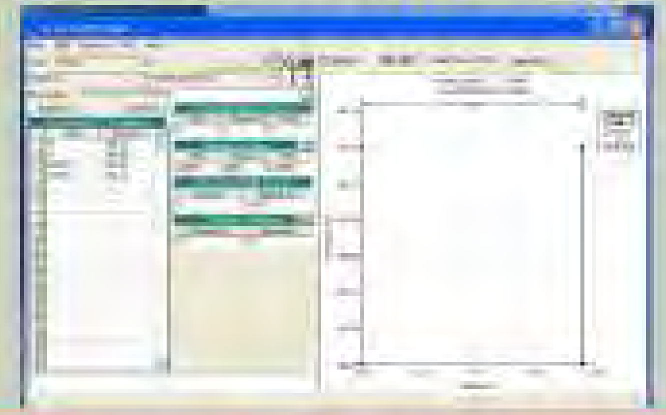
If there were flood plains, this box defines the horizontal right and left limits of the chainages of the main channel (see page after next for example)

Mannings n values
LOB & ROB would be the left and right flood plains respectively

Click on "Apply Data" to add the cross-section to the model and to view the section

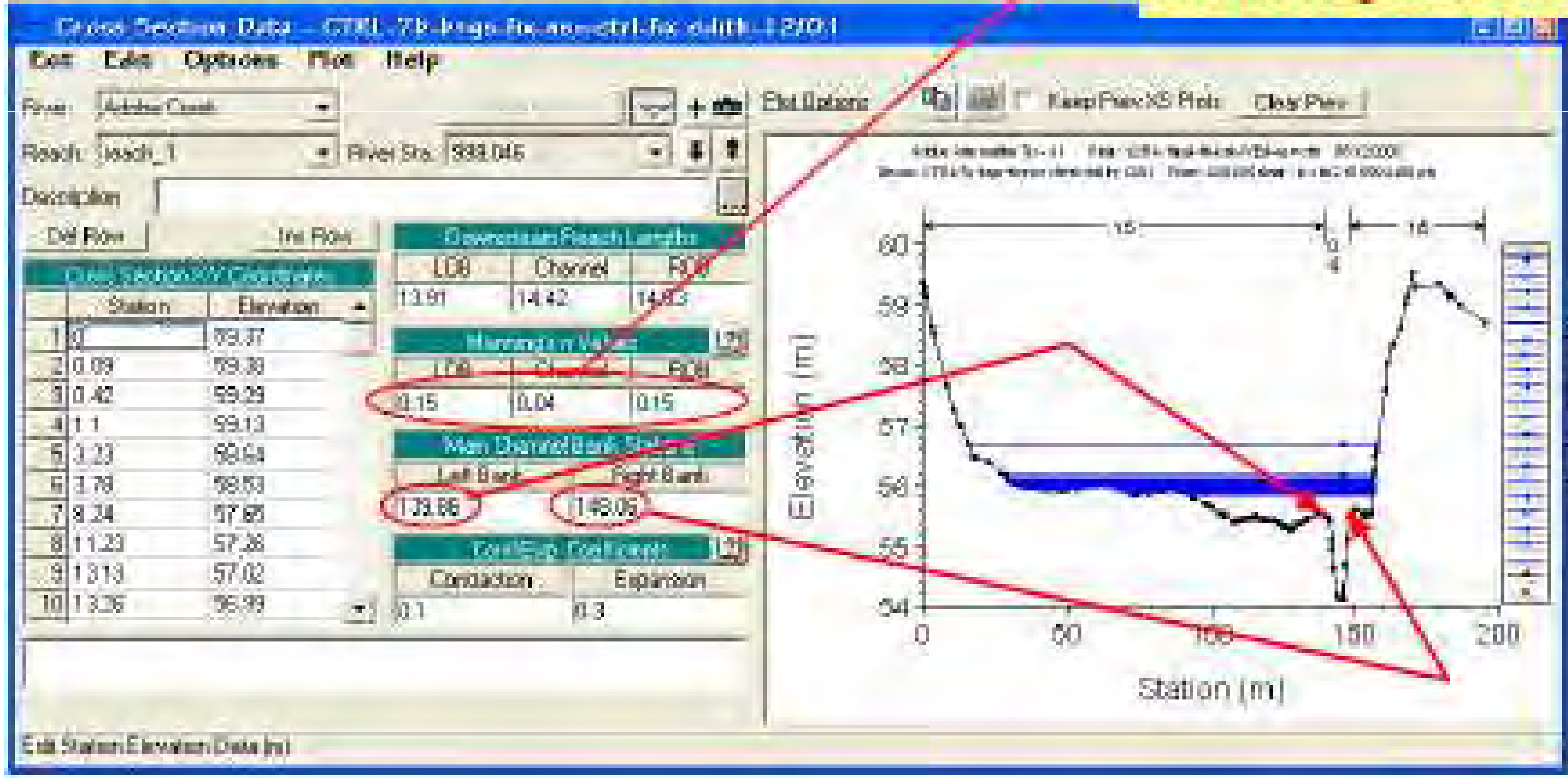


Main Channel Chainage Definition



Flume Example

Note the different Manning's n values in the channel and on the right and left flood plains for a river.



Adobe Creek Example

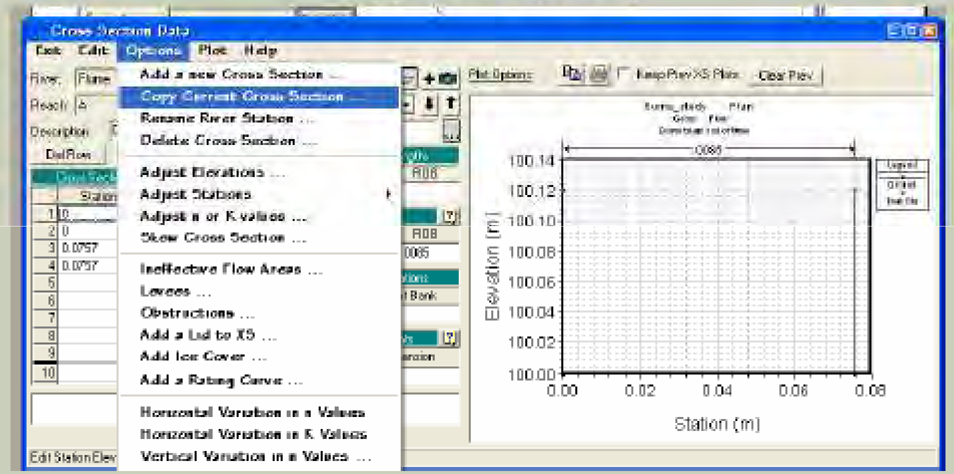
Since the geometry is uniform from the upstream to downstream extent, we can make use of the cross section interpolation tool to compute the geometry with the specified cross section spacing

This will take

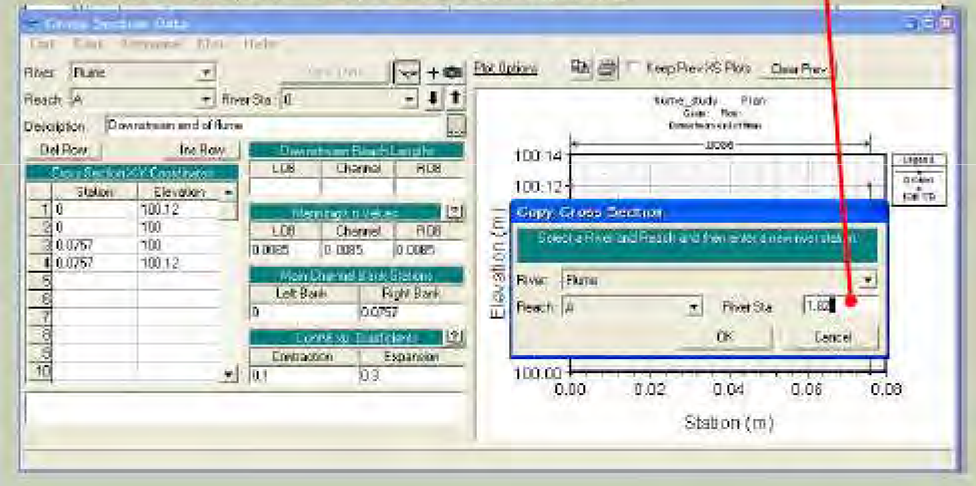
a few steps….

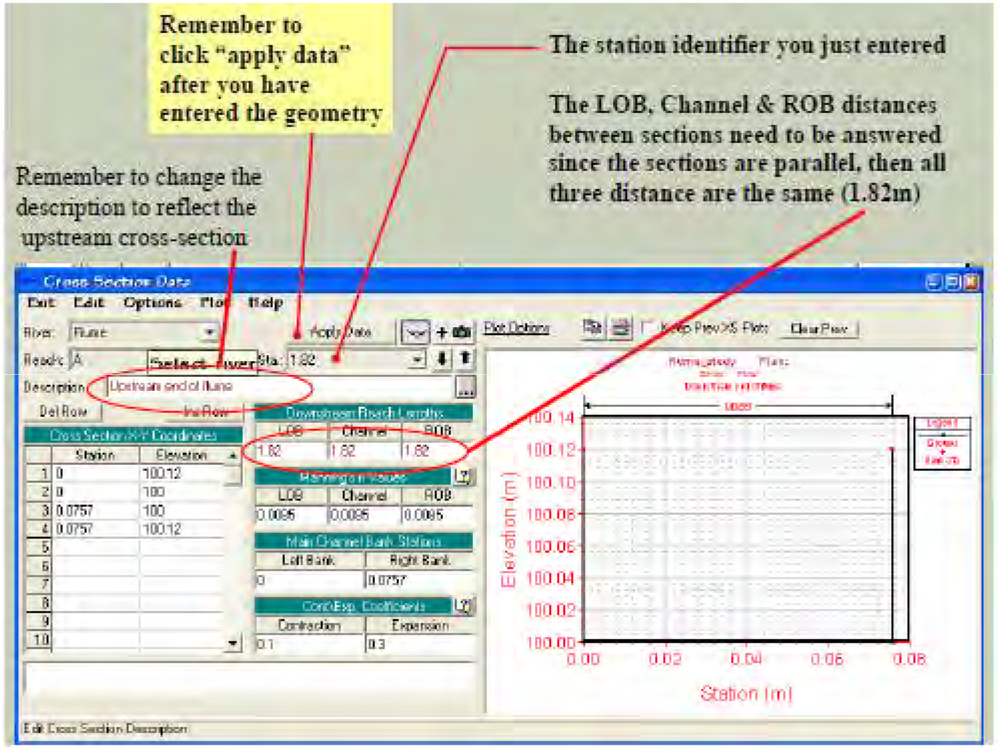
Add a new cross section at the upstream end river station 1.82 m

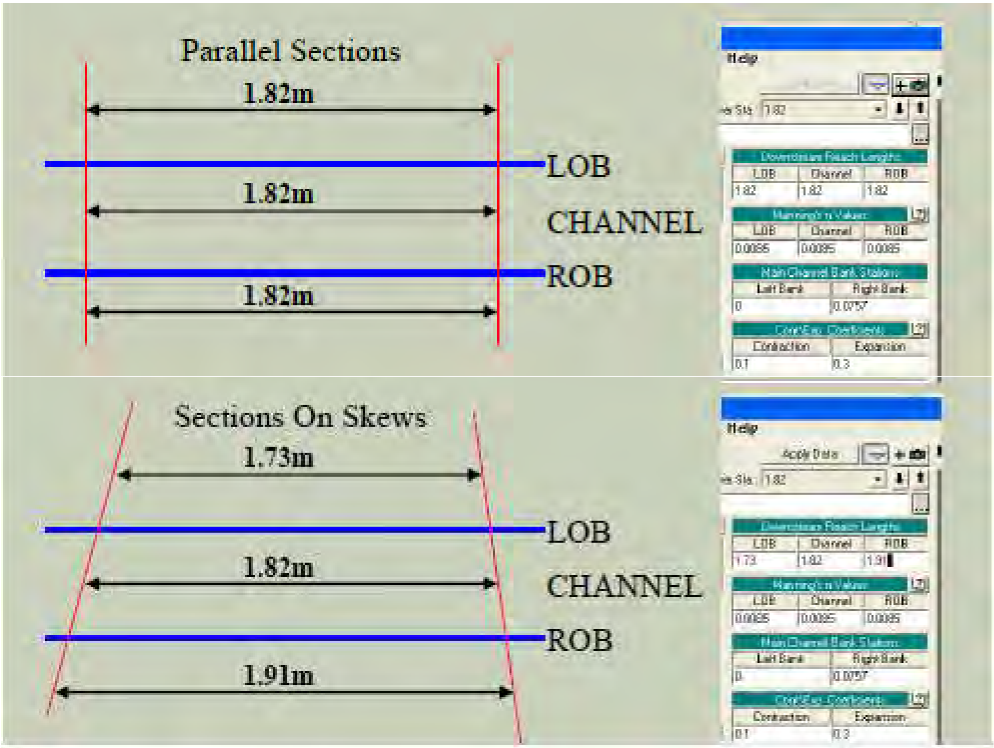
(Since we have the same geometry we are going to make use of the copy section function)



Enter the station chainage at the upstream end of the flume (for the flume the upstream end will be located at1.82m)
Otherwise, if the geometry was different, you would
Click – options / add a new cross section

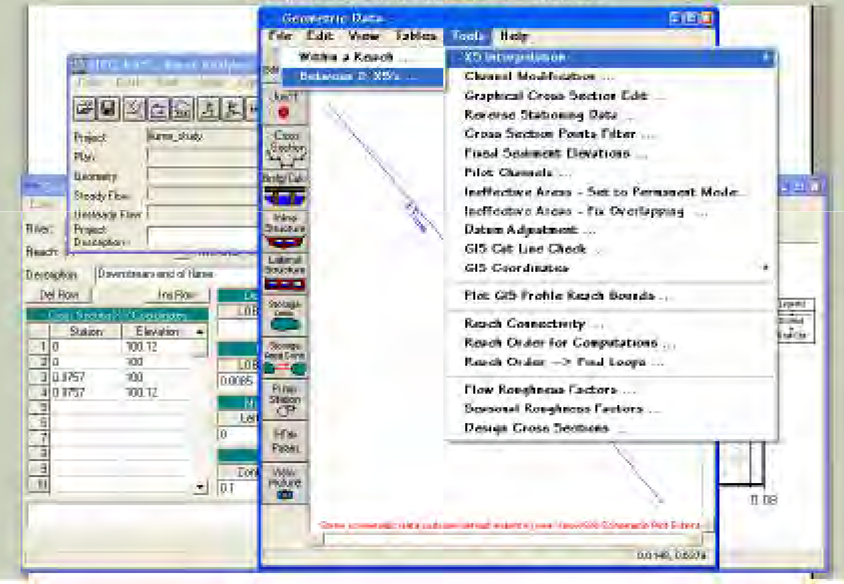



Remember to click "apply data" after you have entered the geometry
The station identifier you just entered
The LOB, Channel & ROB distances between sections need to be answered since the sections are parallel, then all three distance are the same (1.82m)
Remember to change the description to reflect the upstream cross-section
Cross Section Data
Exit Edit Options Plot Help
River: Flume
Reach: A
Select River
Apply Data
Sta.: 1.82
Plot Options
Keep Prev XS Plots
Clear Prev
Description: Upstream end of flume
Del Row
Ins Row
Cross Section X-Y Coordinates
Station Elevation
1 0 100.12
2 0 100
3 0.0757 100
4 0.0757 100.12
Downstream Reach Lengths
LOB Channel ROB
1.82 1.82 1.82
Manning's n Values
LOB Channel ROB
0.0095 0.0095 0.0095
Main Channel Bank Stations
Left Bank Right Bank
0 0.0757
Cont\Exp Coefficients
Contraction Expansion
0.1 0.3
Elevation (m)
Station (m)
Edit Cross Section Description



Parallel Sections
1.82m
LOB
1.82m
CHANNEL
ROB
1.82m
Sections On Skews
1.73m
LOB
1.82m
CHANNEL
ROB
1.91m

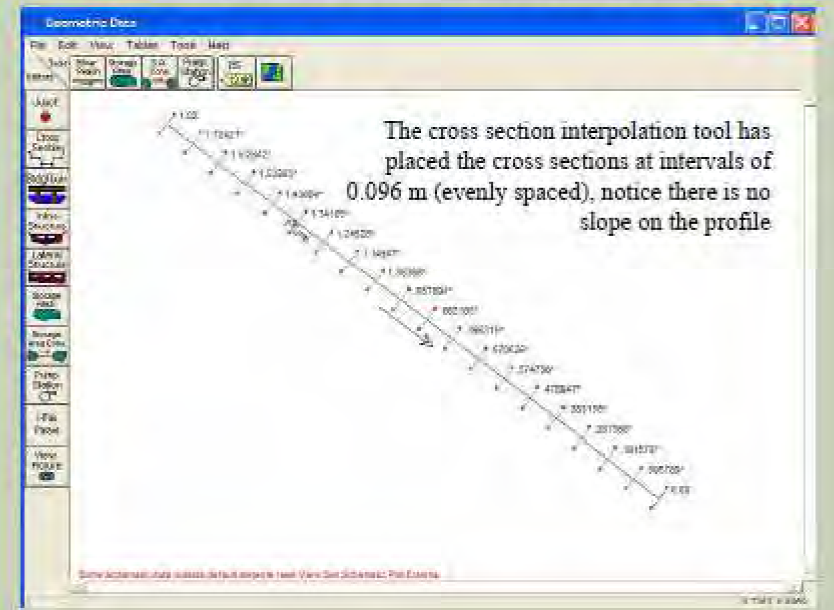
Now it's time to interpolate cross sections…In the main geometry menu click on tools/XS interpolation and select between two cross-sections



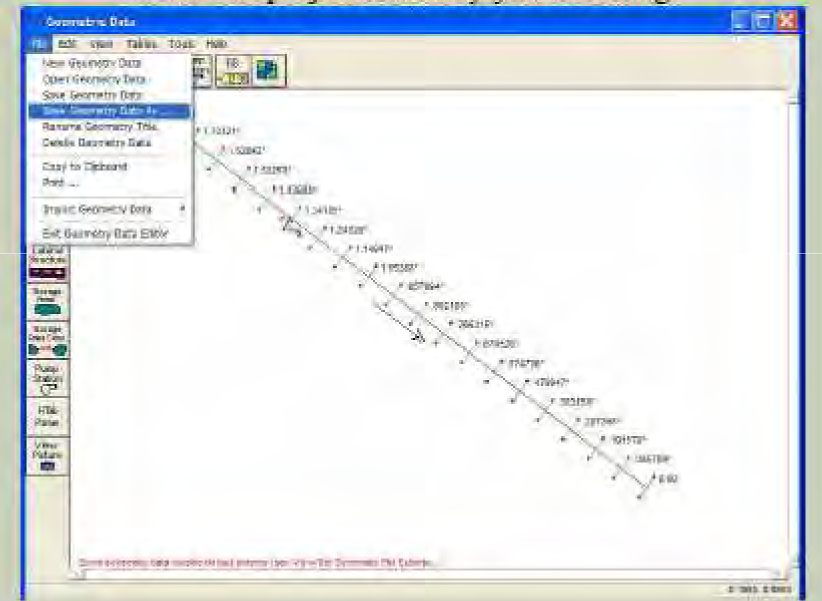
Enter 0.1 m as the max distance between XS's, then hit the interpolate button and your geometry is complete

If you continued to have different cross sectional geometry at each cross-section, you would continue to add new cross-sections and enter the distances between each section.

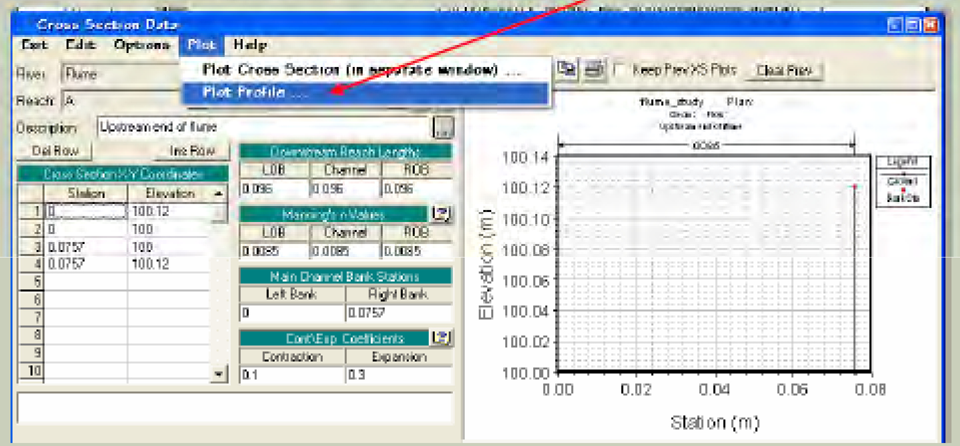
Your main geometry menu should now look like this

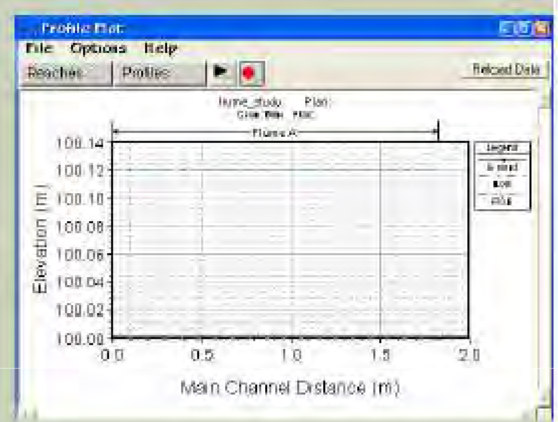


Now is a good time to save your geometry data, remember the .g file? Name and save it in the Ras project directory you are using.



Let's look at the profile for our model, click on the cross section data menu tab, then plot/profile





If your plot does not look like this:
- go to "options" on the Cross section data window
- go to "scaling"
- change y axis min to 100.0
- change y axis max to 100.14
- change y axis increment to 0.02
- click O.K.
- go to "options / variables"
- click on the first two that are un checked (left main channel bank elevation & right main channel bank elevation), click O.K.
- click "options /grid" and make sure that "thick border, major and minor tick grids" are all on.

Now let's put some water in this flume…

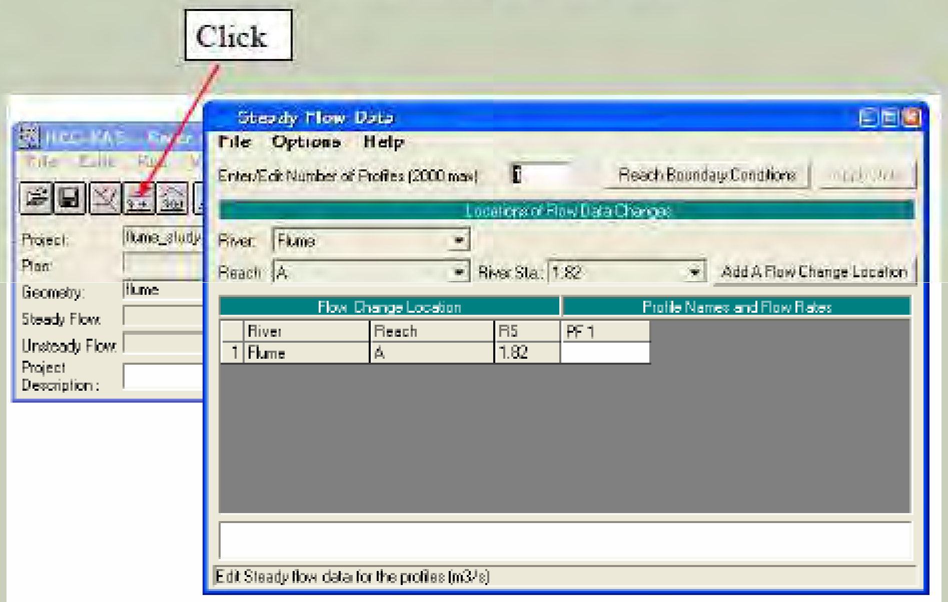
Flow data (.f files)

- Steady (constant with time)
- Unsteady (varies with time)
- Regimes(supercritical, subcritical, mixed)
- Boundary conditions:
  1. Supercritical-upstream
  2. Subcritical-downstream
  3. Mixed-both

Flow data continued

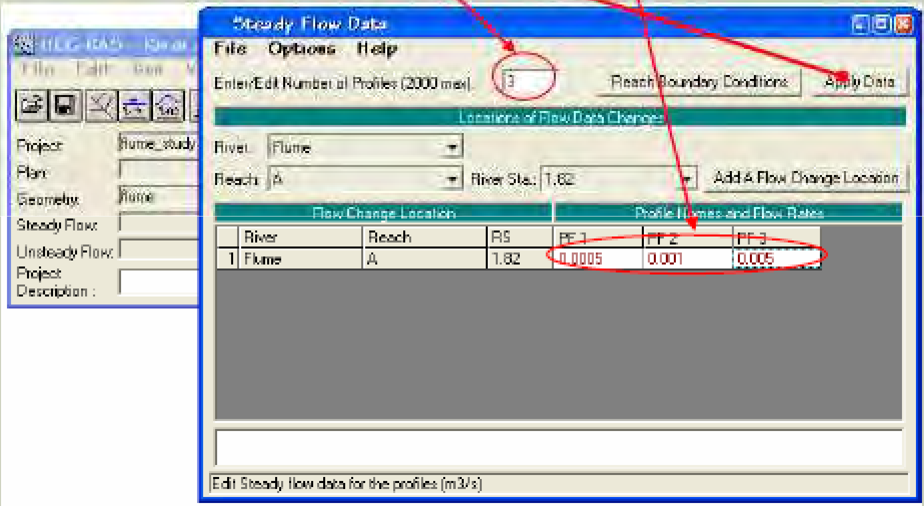
- Boundary condition types:
  1. Known water surface elevation
  2. Critical depth- free overfall or weir
  3. Normal depth- downstream energy slope required
  4. Rating curve (WSE as a function of discharge)

Now open the steady flow data menu

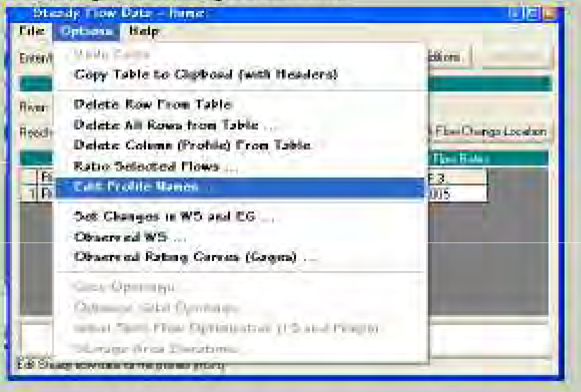


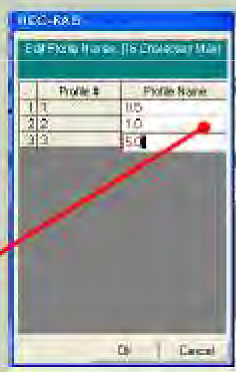
Let's consider 3 flows 0.5,1.0, and 5 liter/s

Enter and apply the data



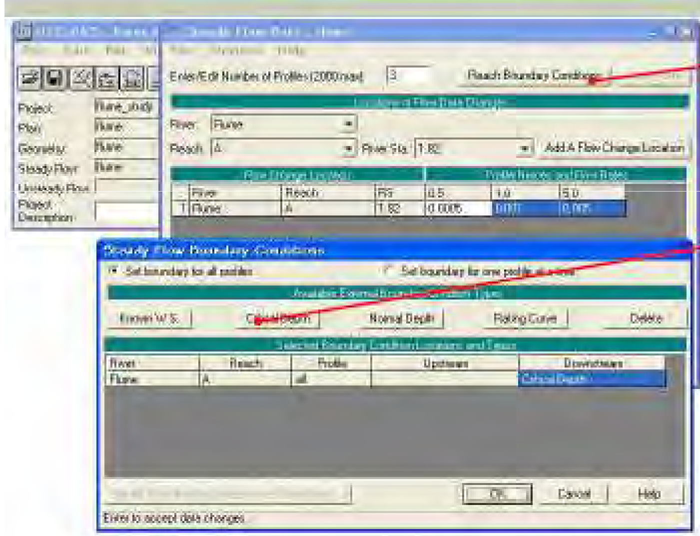
Now we are going to change the profile names (from PF 1, PF2, PF3)
-On the "steady flow data window"
- Chose options / edit profile names





Double click each box on the right of the window to what you see in the HEC-RAS box, and then click O.K.

Now we need to specify the boundary condition, let's assume subcritical flow and critical depth at the overfall therefore a downstream boundary condition and a H2 profile



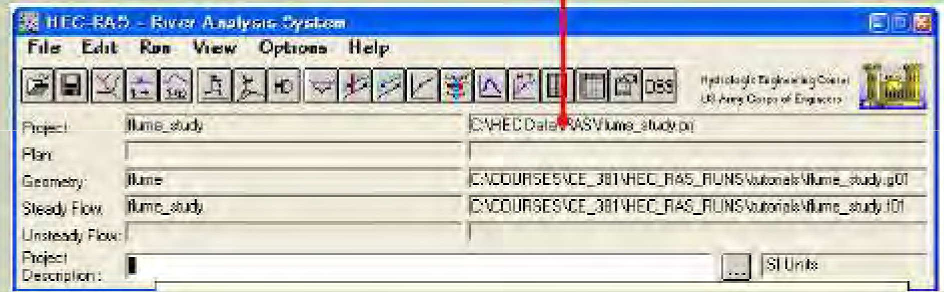
Select Reach Boundary conditions from the steady flow menu,

then select critical depth in the boundary conditions menu

Easy enough now save the data to your directory, this creates a .f file. Since this is the first time saving the flow data use the "SAVE FLOW DATA AS" option

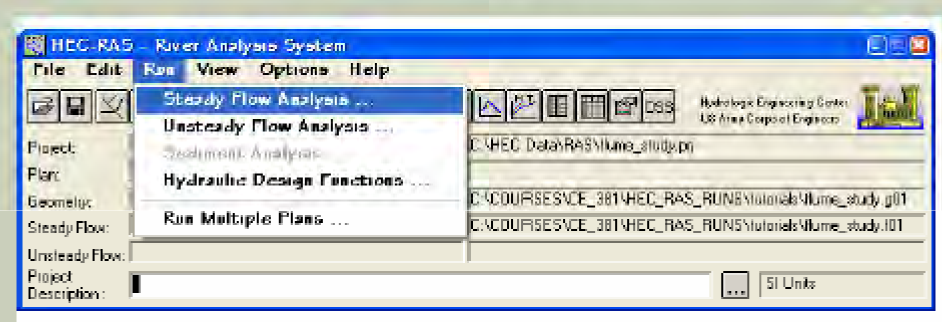
Your main HEC-RAS interface should show the following now

MAKE SURE THAT ALL OF YOUR FILES ARE STORED IN THE SAME DIRECTORY (what ever that directory may be) OR, THE SIMULATION WILL NOT WORK

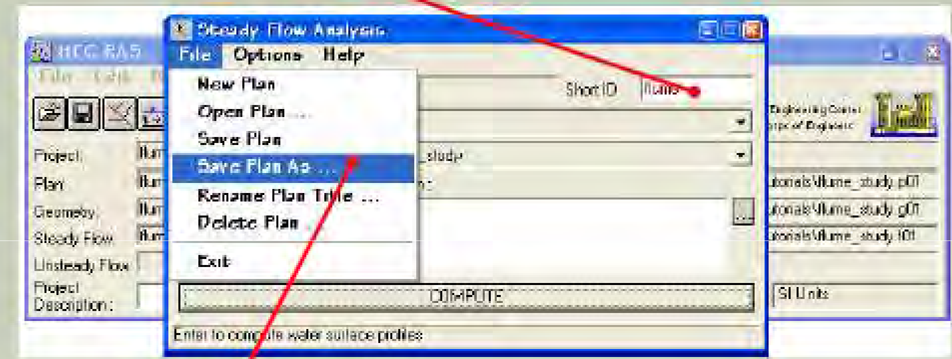


One last step before we run the simulation, we must couple the geometry and flow with the plan (.p file)

On the main Hec-Ras interface select Run and steady flow analysis



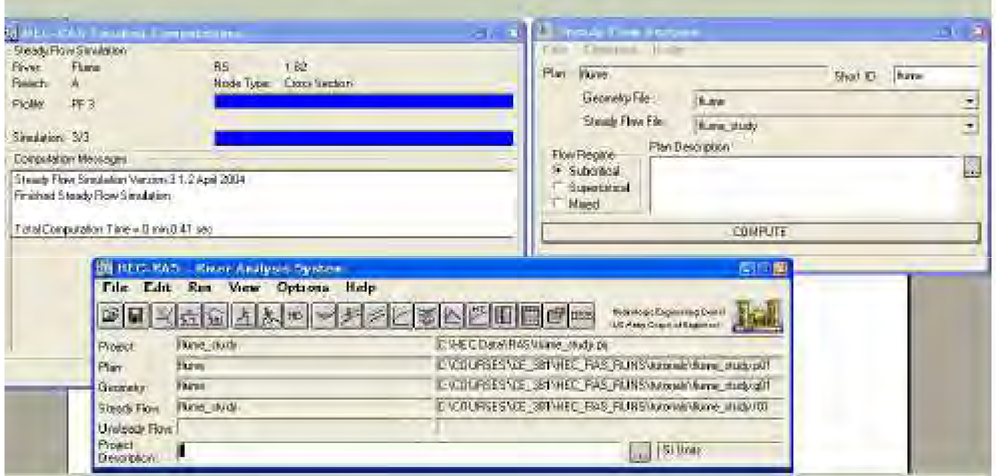
Go ahead and name the plan and save it in the working directory and give a short ID, maybe "flume", use the same name in the station identifier box



Now we can go ahead and save a plan with the existing geometry and flow data

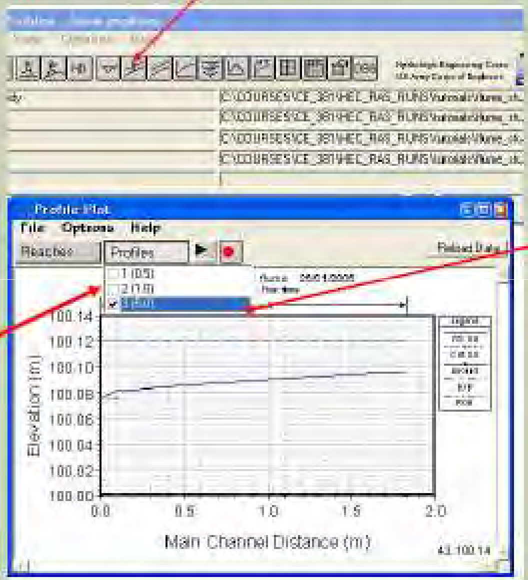
Now we are ready to Run our simulation, hit the "compute" button when ready

When the following screens are visible the simulation has been run



Close the computation screen and congratulations you just ran a HEC-RAS model!

Let's look at the water surface profile generated for the 5.0 L/s flow,click on the view profiles tab on the main interface

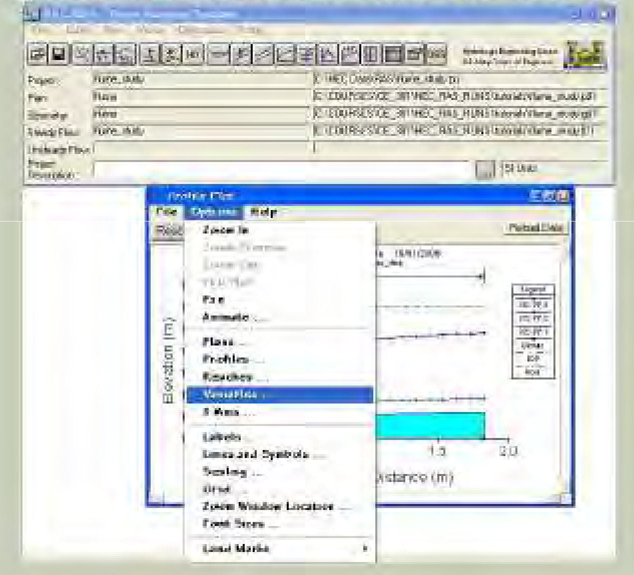


Make sure the other profiles are unchecked

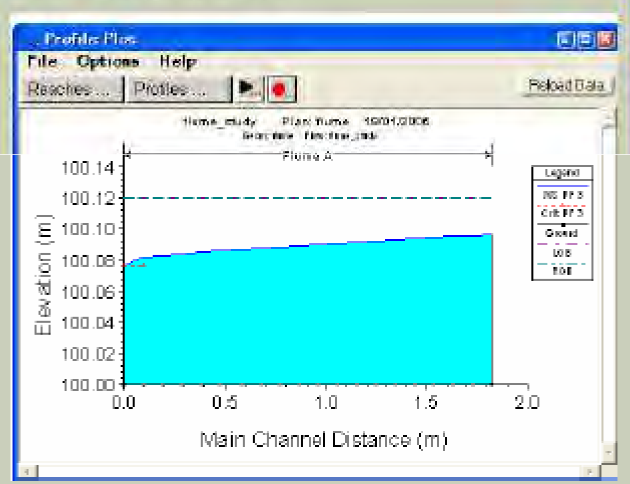
A series of variables can be plotted on the profile.

Lets add the critical flow depth….

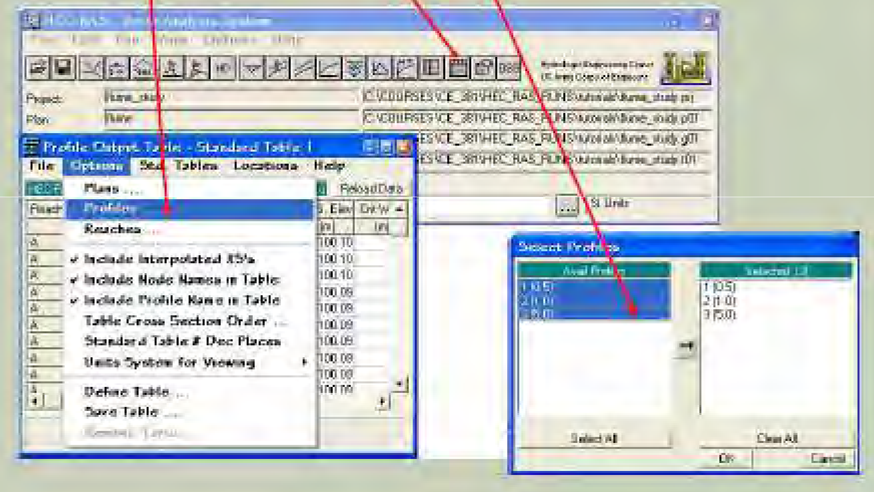
Navigate to options / variables on the Profile Plot Window



Make sure that the following are checked on:
-Left main channel bank elevations
- Right main channel bank elevations
- Filled in water surface
- Critical depth elevation

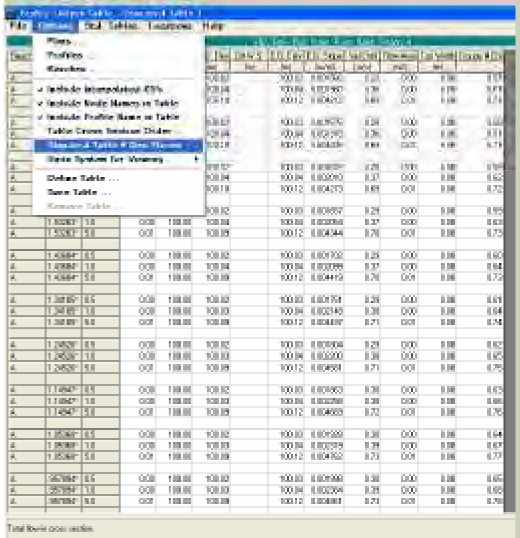


Click on the tables tab of the main menu interface.Select options/profiles, select all three flows and click the O.K. button.

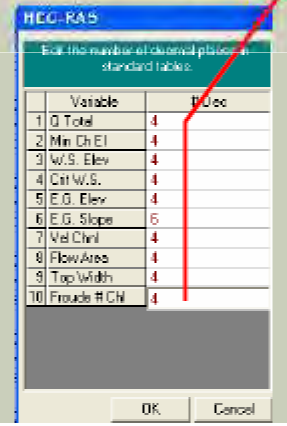


The standard output of HEC RAS lists variables with several different levels of precision, which is dependent upon the particular variable. Since the flume is a very small representation of a "river reach", we need to increase the level of precision on the output to properly capture the scale of the flume.

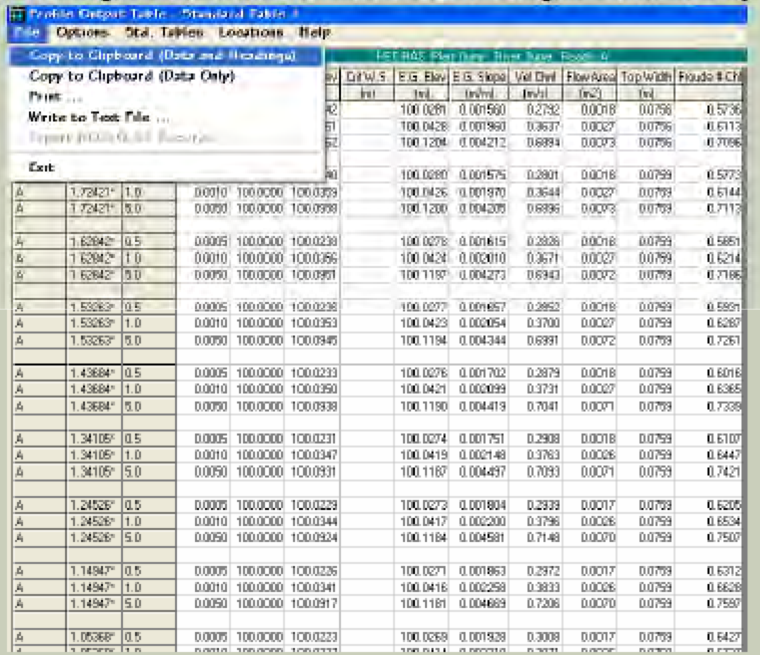
On the "PROFILE OUTPUT TABLE", click on OPTIONS / STANDARD TABLE & DEC. PLACES.



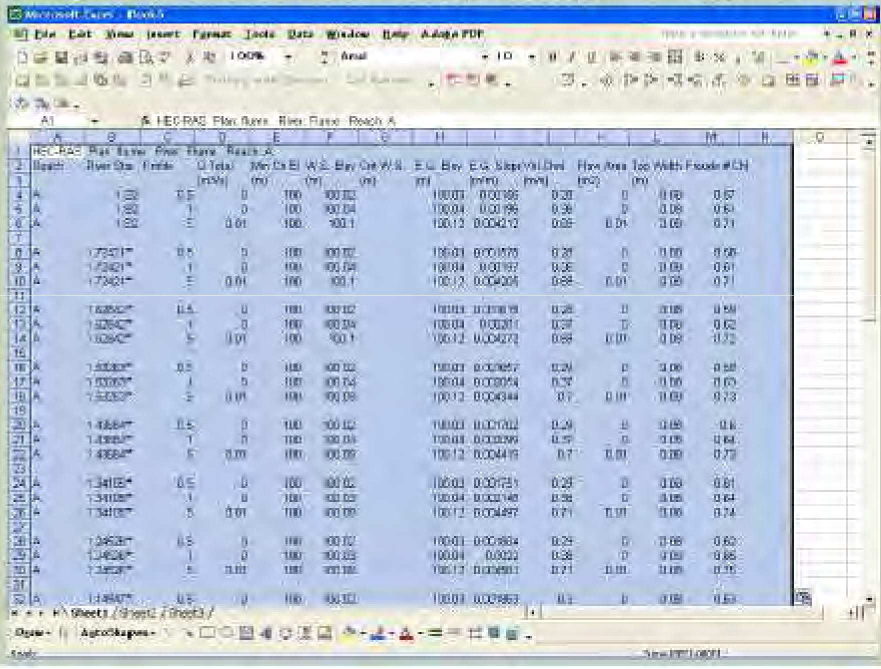
Change any # Dec. values less than 4 to 4 decimal places



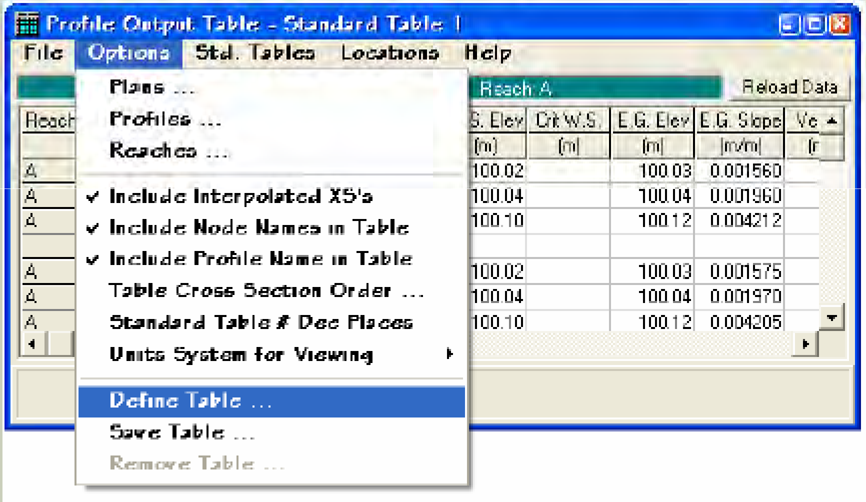
You can export the data from the Profile Output Table to any application



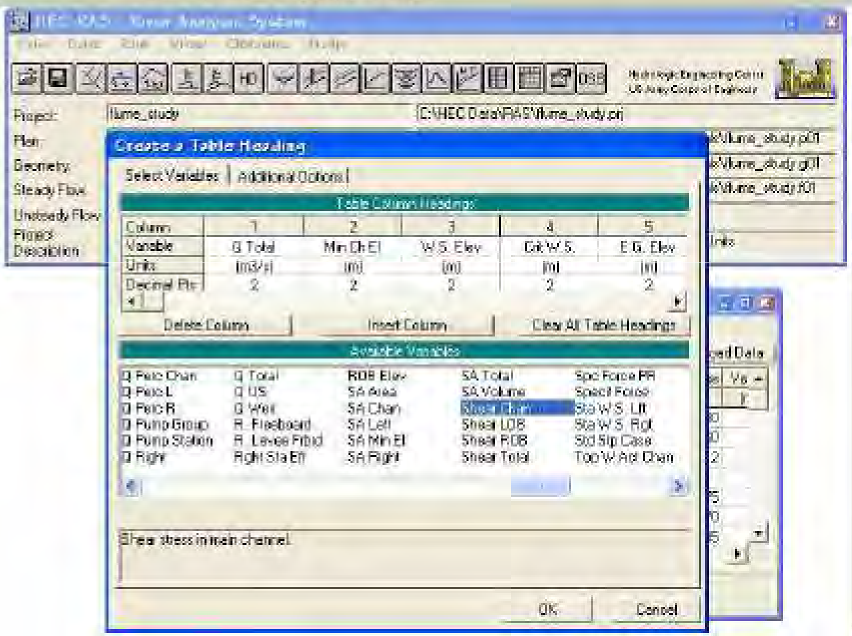
And paste the data into excel or any other applicable application



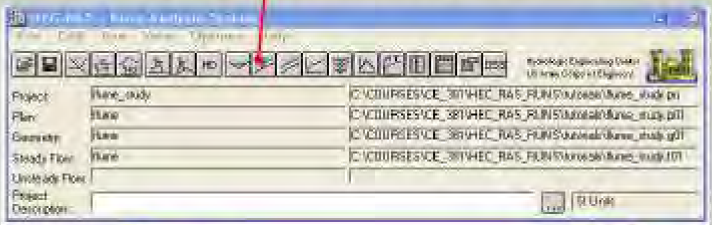
This table provides water surface elevation, velocity, Froude number, area, width, energy grade line etc. Many more options are available under options/define table

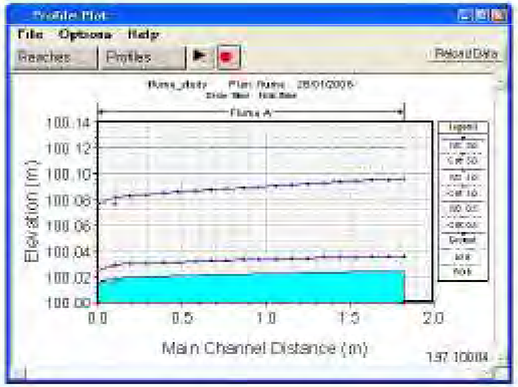


Double click any parameter (in the Available Variables) to add it to your table.

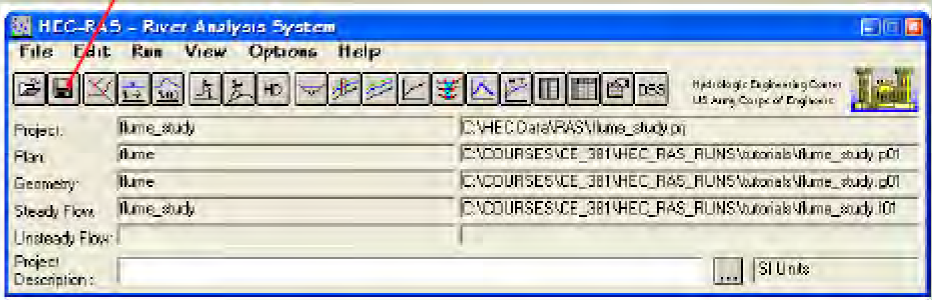


Now that we have selected all three flows, we can go back to the main menu and the plot profile tab to compare the three flows.





One last step let's go ahead and save the .prj or project file we have been working on.



Nice work, you now know how to run HEC-RAS.

When you attempt to calibrate your results to observed data, there are only three things you can vary:

A. the Manning's n coefficient (within a reasonable range)
B. flow range (examine the min, max and mean flow ranges that you observed)
C. the exact location of the critical depth location