2D Sediment Workshop

1 Objective

In this workshop, you will use HEC-RAS to setup a simple 2D sediment simulation.

2 Introduction

The workshop is based on a series of laboratory experiments conducted by Weise (2002). The experiments were designed to study contraction scour and expansion deposition of non-uniformly sized sediment. The experiment consists of a 16.5-m long rectangular flume with a variable with between 0.5 m and 1 m. The straight side of the flume is a smooth glass while the curved side with rough concrete. The initial bed had a 20-cm layer of fine gravel with a 5.5 mm mean diameter and geometric standard deviation 1.47. The upstream flow was 0.15 m^3/s and the downstream water depth was 0.312 m. The experiment duration was 125 min.

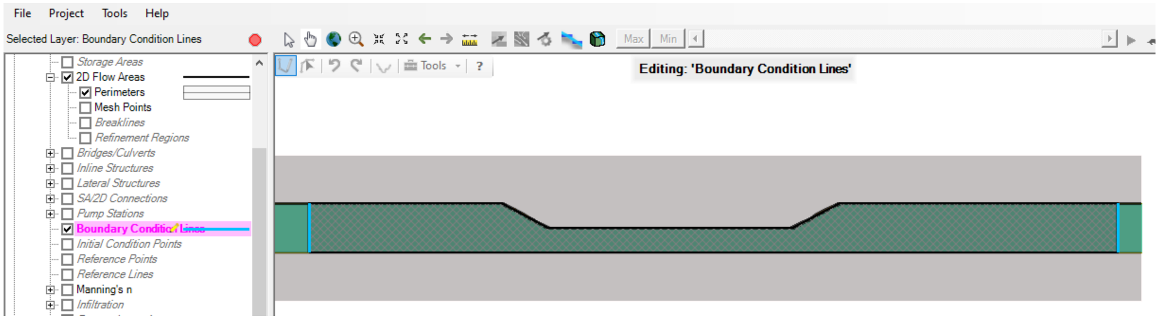
3 Workshop Instructions

3.1 Open the RAS Project

The HEC-RAS project is named Weise.prj and should be contained in a folder names 2D Sediment Workshop.

3.2 Geometry

The workshop already contains a terrain, computational mesh, boundary condition lines and Manning's n layer. The grid resolution utilized for this workshop is relatively coarse but is utilized for computational efficiency. It is always recommended to start with a coarse grid resolution to get the model running and narrow down on the model parameters and settings, and then increase the model resolution before doing production runs. Sediment transport has a lot of options and parameters to calibrate, so being able to speed up the initial exploratory calibration runs is very useful. Once the calibration options and parameters have been narrowed down, the resolution can be increased to finalize the calibration and do the validation runs. For this model, Manning's polygons have been defined in order to specify different roughness coefficients for the flume walls. The roughness is utilized in the hydrodynamic model at faces while in sediment at cells. Because the model only uses a single value extracted from the cell centroids and face centers, it is important the Manning's polygons extend into the first cell/face right next to the walls.



3.3 Flow Data

Open the Unsteady Flow editor. The boundary condition lines have already been defined as Upstream and Downstream.

Select Location in table then select Boundary Condition Type

	River	Reach	RS	Boundary Condition

	Storage/2D Flow Areas		Boundary Condition
1	2DArea	BCLine: Downstream	
2	2DArea	BCLine: Upstream	

Click on the **Downstream** BCLine and select Stage Hydrograph.

Set a constant downstream stage at 0.312 m

(Note: you can set a constant downstream stage by repeating the stage – or flow in the next step – in the time series.)

It is usually best practice in 2D modeling to **Use Initial Stage** with a downstream stage boundary condition. ☑ Use Initial Stage (recommended)

Click on **Upstream** BCLine and select Flow Hydrograph.

Set the flow to 0.15 m3/s

Also set the **EG Slope** to 0.001 for distributing flow along BC Line and select **TW Check**.

EG Slope for distributing flow along BC Line: 0.001 ☑ TW Check

Plot Data | OK | Cancel

It is not necessary to set the initial water surface since the model will use the downstream stage to initialize the model.

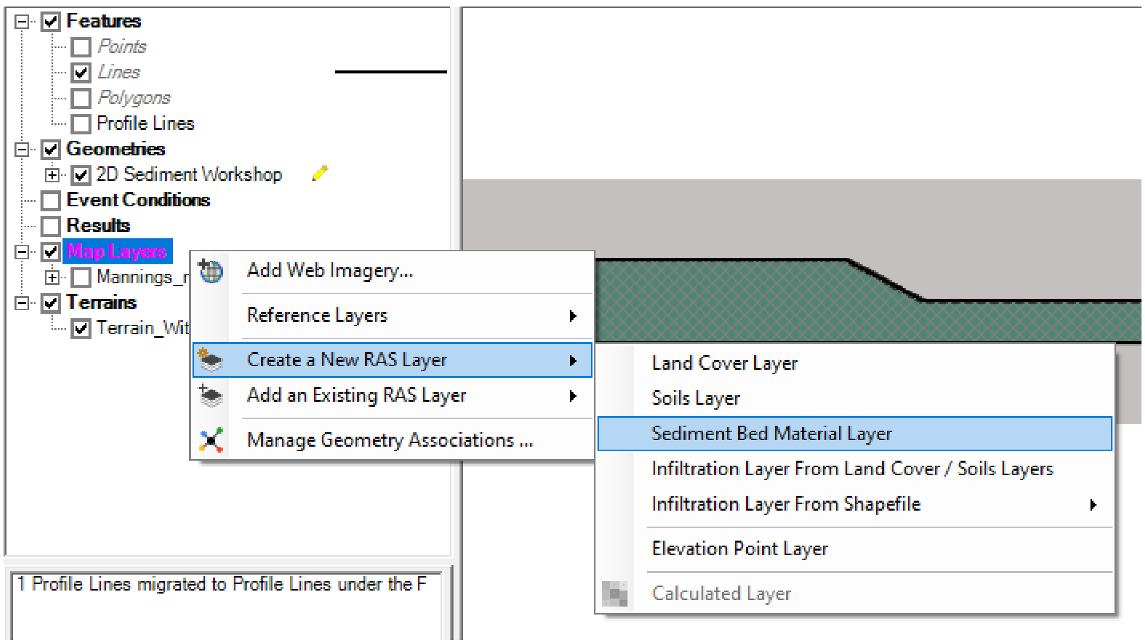
Save As and give it a name (e.g. "2D Sediment")

3.4 Sediment Data

3.4.1 Define Bed Gradation Polygon in Mapper

Before you define sediment data, it is useful to define the spatial distribution of your bed gradations in Mapper (so you can populate them with gradation data in the sediment editor).

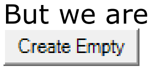
Open Mapper, and Right Click on **Map Layers**. Select **Create a New RAS Layer→Sediment Bed Material Layer**.



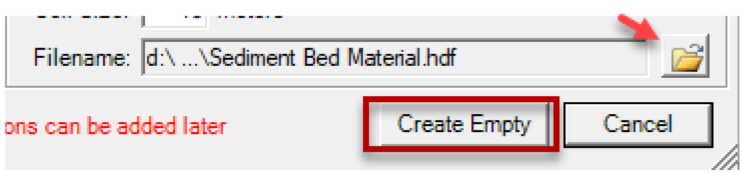
You can press **No** to the question about spatial reference system.

This editor was developed to import layer data from raster or shape files.

But we are just going to draw a singly polygon, so you will press the **Create Empty** button.

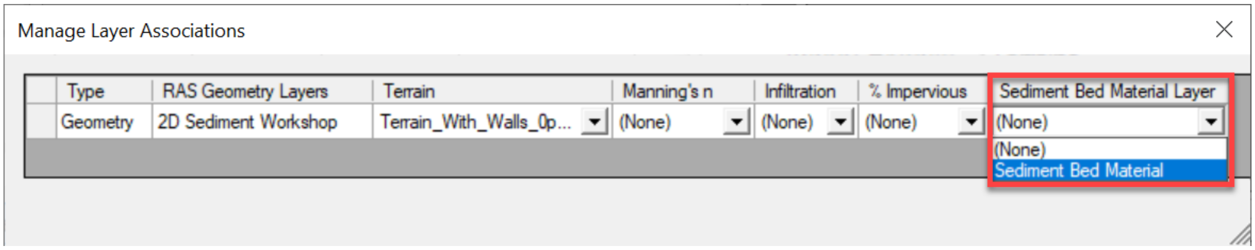


Press the folder if you want to change the name to something other than the default "Sediment Bed Material."



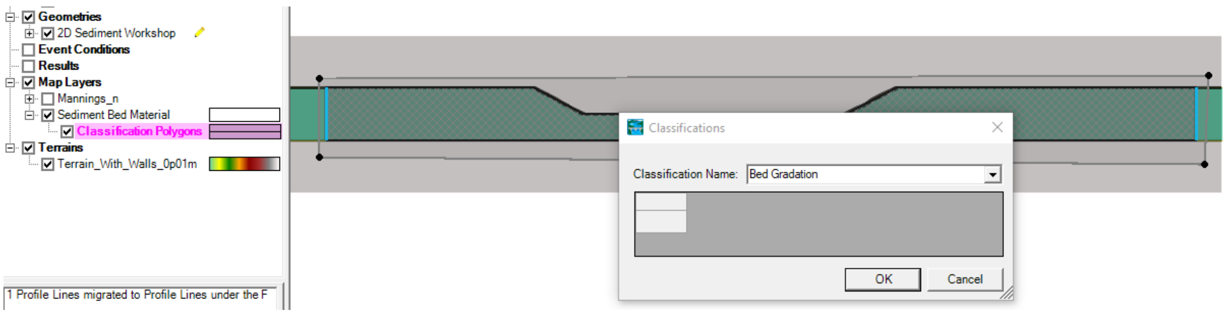
Expand the Association Editor that comes up until you can see the **Sediment Bed Material Layer** dropdown menu.

Select the empty Sediment Bed Material layer you just created.



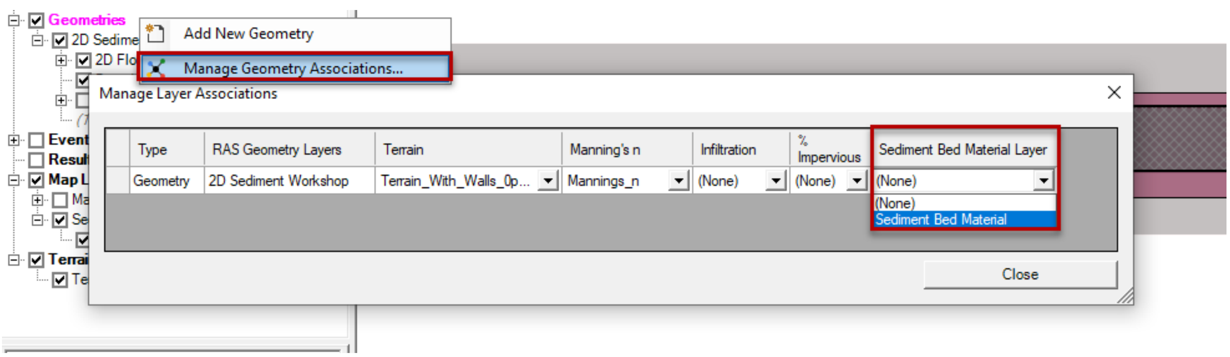
Start editing the **Classification Polygons** in this new layer, choose the draw tool , and draw a polygon around the mesh.

Give the Polygon a **Classification Name** (e.g. "Bed Gradation").



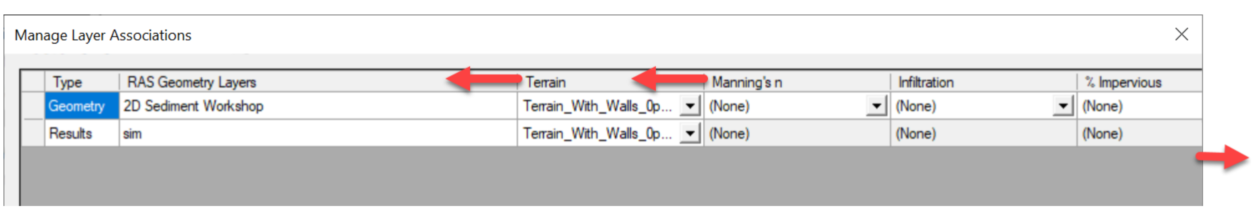
Stop Editing  and save the classification.

Associate this Bed Material Classification with the geometry file by right clicking on the **Geometries** tree node and selecting **Manage Geometry Association**.



Warning Current versions of RAS have a glitch that can make this step difficult.

The Layer Association Manager opens too small so you cannot see or scroll to the Sediment Bed Material Layer. Drag the right edge to expand the editor and see the last column or make the column headers thinner until you see all the columns.



Then the Sediment Bed Material Layer selection does not always save.

Reopen the Geometry Association Manager to make sure if it is selected.

If it isn't – reselect it, and then click on the Terrain drop down.

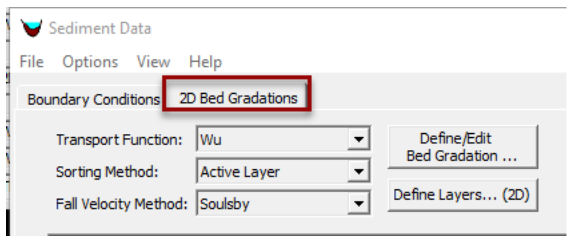
Reselecting the current terrain should get it to save.

3.4.2 Basic Options

Open Sediment Data editor by clicking on the icon 

from the main RAS window.

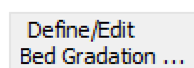
This window opens to the **Boundary Conditions** Tab, click on **2D Bed Gradations** to start there.



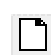
Set the **Transport Function** to **Wu**, the **Sorting Method** to **Active Layer**, and the **Fall Velocity Method** to **Soulsby** as shown in the screenshot below.

Notes: The only **Sorting Method** supported by the 2D sediment model is the **Active Layer**. Choosing any other sorting method will default back to the **Active Layer** method.

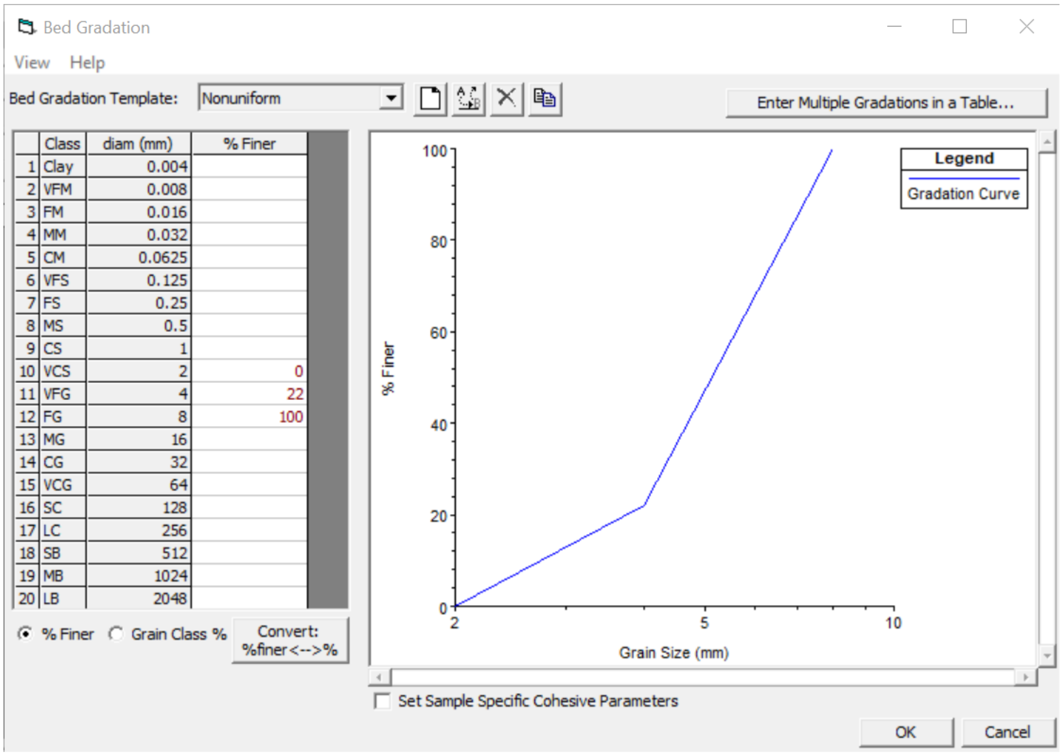
3.4.3 Bed Gradation

From the **Sediment Data** editor press 

.

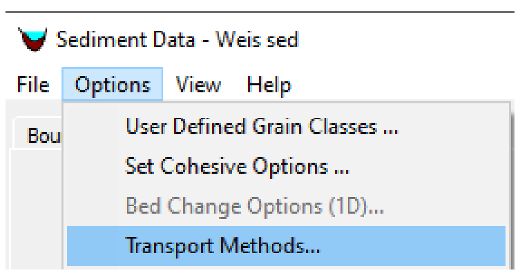
Create a new **Bed Gradation** by pressing the "New" button . Give it a name (e.g. "Nonuniform"). Enter the gradation information as shown in the figure below.

Since the grain size distribution is moderately well sorted, the fractions are contained within only two grain classes. This is relative few. One of the improvements which could be done to the model is to specify User-defined grain classes with more resolution in the range utilized in the experiment, so that more grain classes are utilized.

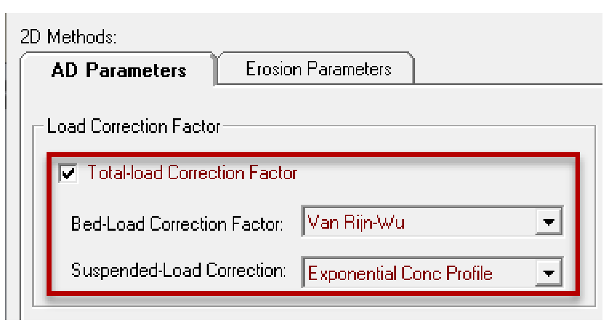


3.4.4 *Transport Method Options*

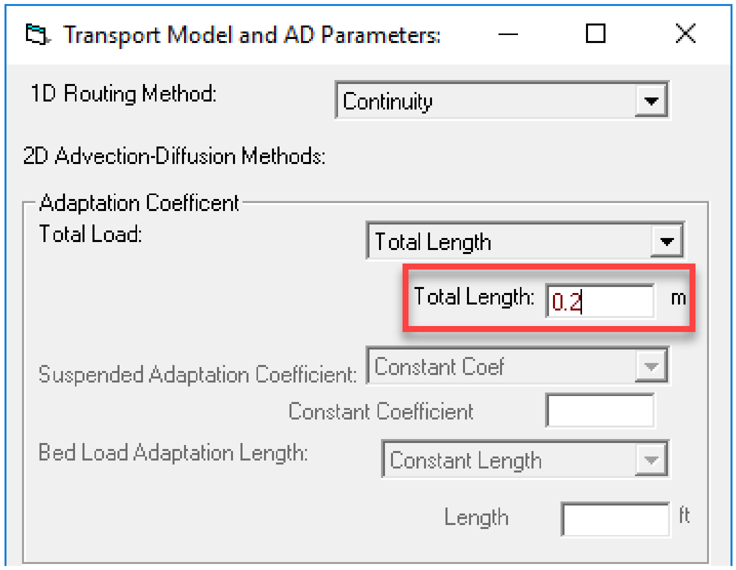
From the **Sediment Data** editor, click on **Options** | **Transport Methods…**



Under **AD Parameters**, set **the Load Correction Factor** Options.



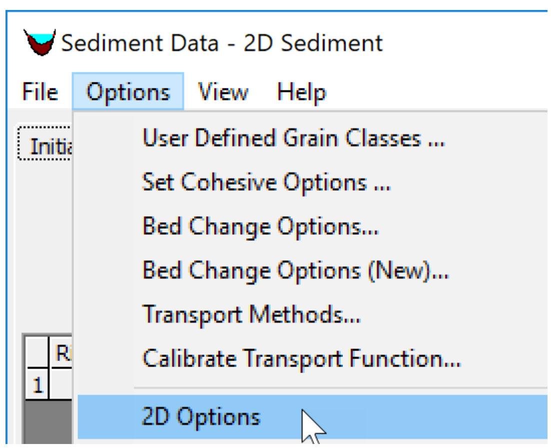
In the **Adaptation Coefficient** section, set the **Total-load** method to **Total Length** and specify a value in the editor of 0.2 m as shown in the screenshot below.



Click on **OK**, and save your changes in the **Sediment Data** editor.

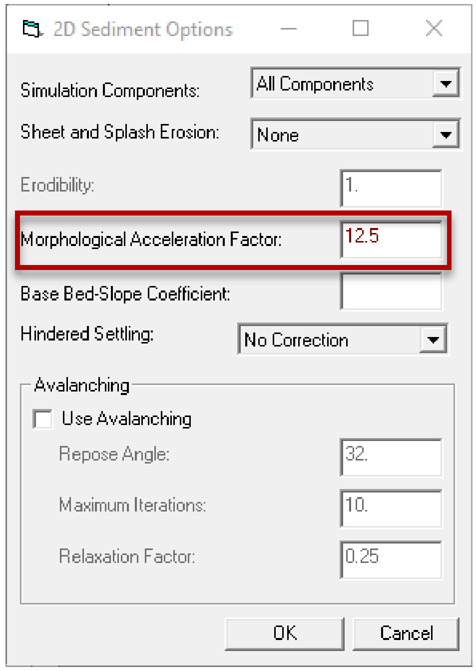
3.4.5 2D Sediment Options

In the **Sediment Data** editor lick on the menu **Options | 2D Options** as shown in the screenshot below.



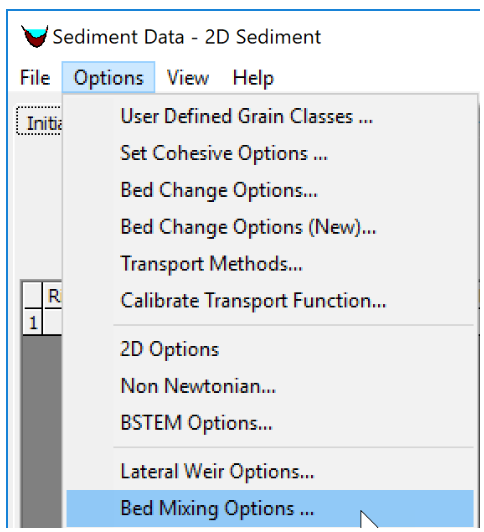
An editor called **2D Sediment Method** will appear. This windows contains only 2D specific sediment options.

In order to speed up the simulation, a **Morphologic Acceleration Factor** of 12.5 will be used. Since the experiment duration is 125 min, the equivalent computational time representing 125 min of simulated bed change is computed as 125 min / 12.5 = 10 min. In general, it is not recommended to use Morphologic Acceleration Factors larger than 20 or 30.

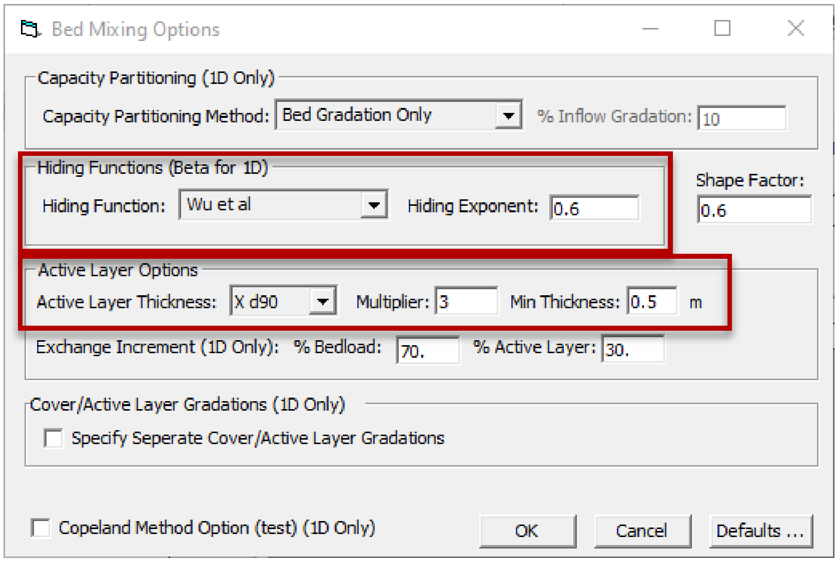


3.4.6 *Bed Mixing Options*

From the **Sediment Data** editor click on the menu **Options | Bed Mixing Options**.



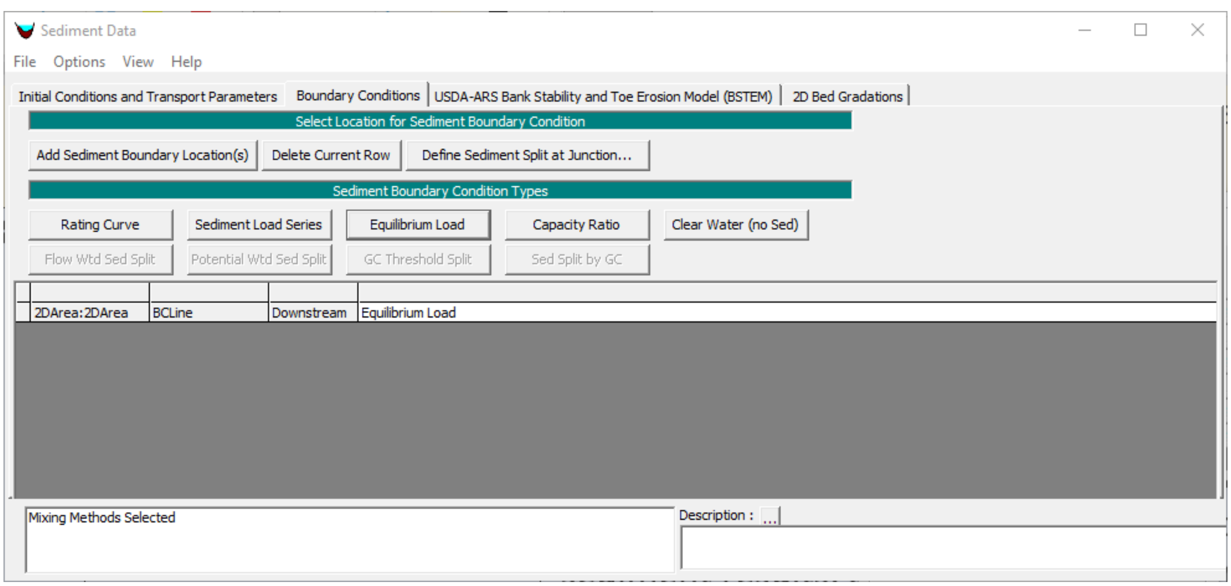
In the **Hiding Functions** section, set the **Hiding Function** to **Wu et al.** the **Hiding Exponent** to 0.6 as shown in the snapshot below.



Click on **OK**, to exit the editor and save your changes.

3.4.7 Sediment Boundary Conditions

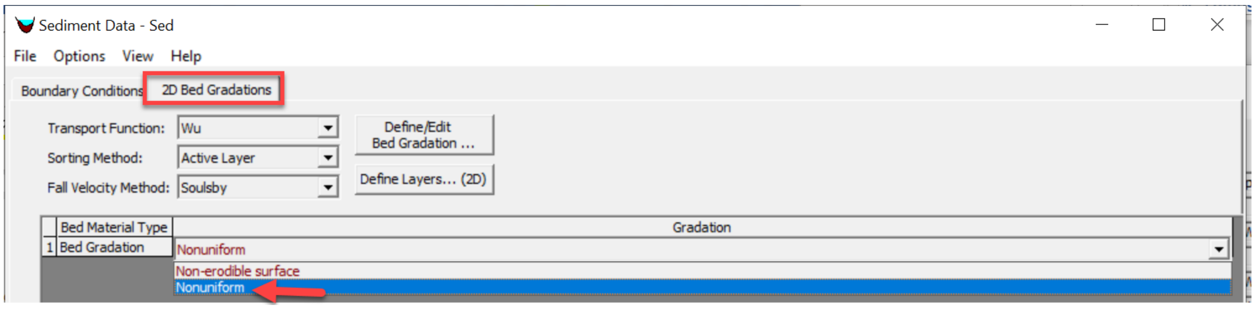
From the **Sediment Data** editor, select the **Boundary Conditions** tab.
The interface automatically searches for boundary condition lines where sediment data could be needed and list them in the table.
By default, any boundaries left unspecified are equilibrium boundary conditions, but this option may also be specified by first select the white entry space and then selecting the **Equilibrium Load** option (see snapshot below).



3.4.8 Select Initial Bed Gradations

Next, select the bed gradations associated with each manning's region.
Open the 2D Sediment Tab.
This will populate a row for each Manning's n region in your geometry file.
This project only has one (base n). Click on the field next to it.
You will get a drop down menu including all of the bed gradations (again, just one in this case) that you defined in Step 3.4.2.

<u>Note</u>: You may have to close the sediment file and open it if you do not see the "Bed Material Type." If you still don't see it, make sure the classification polygon saved and that the mapper association is correct.



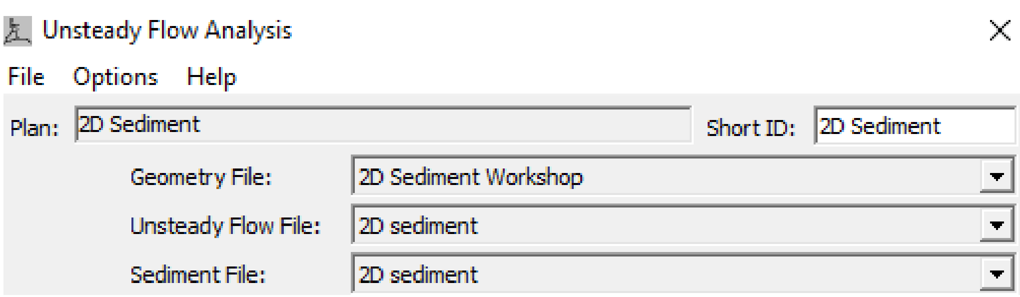
3.5 Unsteady Flow Analysis

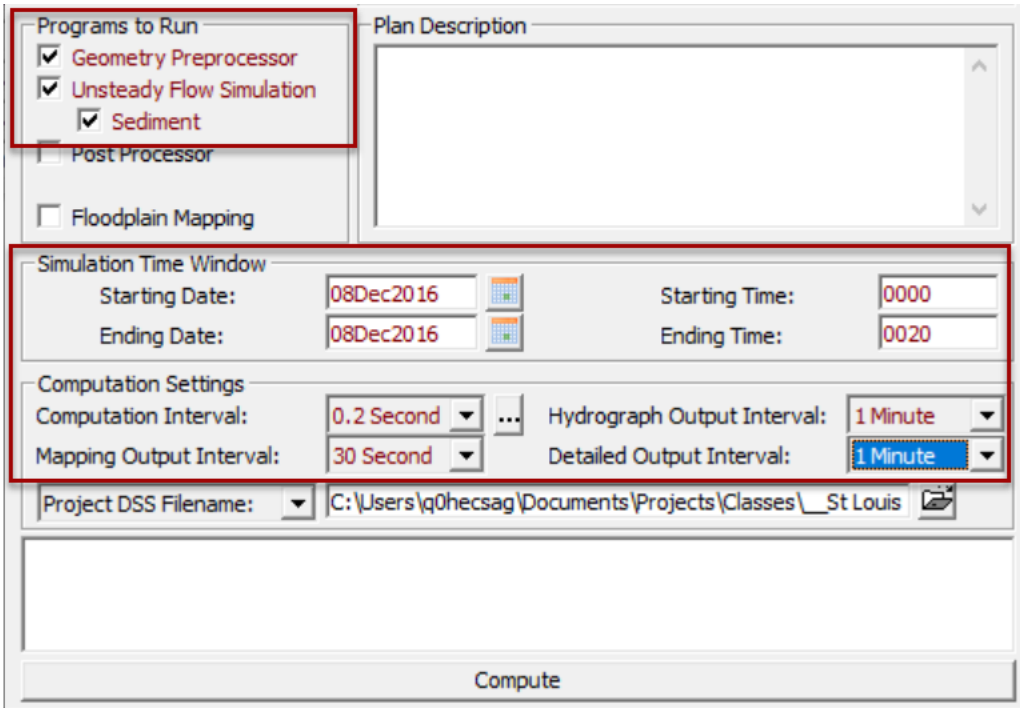
From the main HEC-RAS editor, open the **Unsteady Flow Analysis** editor by clicking on .

Click on the menu **File** | **Save**, and give the plan a name and **Short ID**.

In the section **Programs to Run**, select the checkboxes for **Geometry Preprocessor**, **Unsteady Flow Simulation**, and **Sediment**.

Utilize today's date for starting and ending dates. Start the simulation at 00:00 hours. The experiment duration is 125 min. However, since a morphologic acceleration factor of 12.5 is used, the total simulation duration should be 125 min / 12.5 = 10 min.





3.5.9 Flow Computation Options and Tolerances

In the **Unsteady Flow Data** editor, make the following changes to the **Options→Computation Options and Tolerances** for the **2D Flow Area**.

Because this dataset is a small scale laboratory dataset, the water surface and volume tolerances should be reduced.

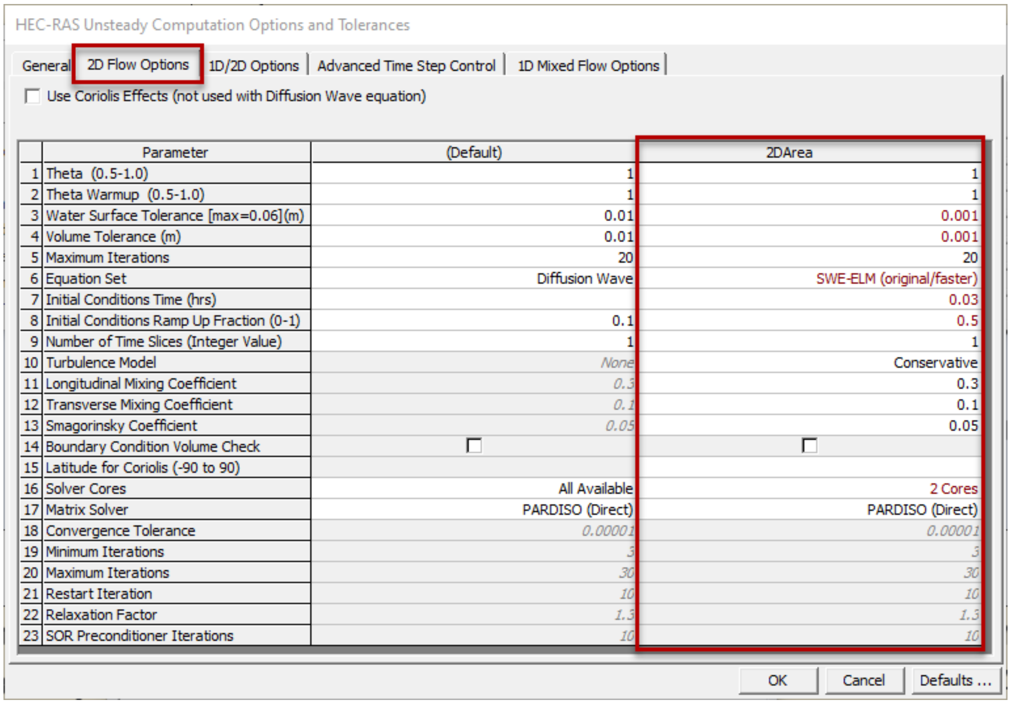
It is important to change the governing equations to the SWE. Use either the ELM or EM solvers.

The other critical thing which the user should turn on is turbulence modeling.

The calibrated turbulence coefficients are provided below.

Since this is a relatively small dataset with only ~2700 cells, there is better to use a small number of cores less or equal to 4 and not the default option of using all of the cores.

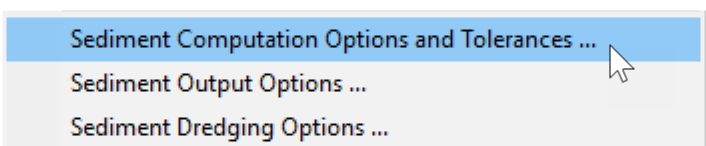
Using too many cores can easily make this dataset run slower than with fewer cores.



	Parameter	(Default)	2DArea
1	Theta (0.5-1.0)	1	1
2	Theta Warmup (0.5-1.0)	1	1
3	Water Surface Tolerance [max=0.06](m)	0.01	0.001
4	Volume Tolerance (m)	0.01	0.001
5	Maximum Iterations	20	20
6	Equation Set	Diffusion Wave	SWE-ELM (original/faster)
7	Initial Conditions Time (hrs)		0.03
8	Initial Conditions Ramp Up Fraction (0-1)	0.1	0.5
9	Number of Time Slices (Integer Value)	1	1
10	Turbulence Model	None	Conservative
11	Longitudinal Mixing Coefficient	0.3	0.3
12	Transverse Mixing Coefficient	0.1	0.1
13	Smagorinsky Coefficient	0.05	0.05
14	Boundary Condition Volume Check	☐	☐
15	Latitude for Coriolis (-90 to 90)		
16	Solver Cores	All Available	2 Cores
17	Matrix Solver	PARDISO (Direct)	PARDISO (Direct)
18	Convergence Tolerance	0.00001	0.00001
19	Minimum Iterations	3	3
20	Maximum Iterations	30	30
21	Restart Iteration	10	10
22	Relaxation Factor	1.3	1.3
23	SOR Preconditioner Iterations	10	10

3.5.10 Sediment Computation Options and Tolerances

In the **Unsteady Flow Analysis** editor, select the menu **Options** | **Sediment Computation Options and Tolerances**.



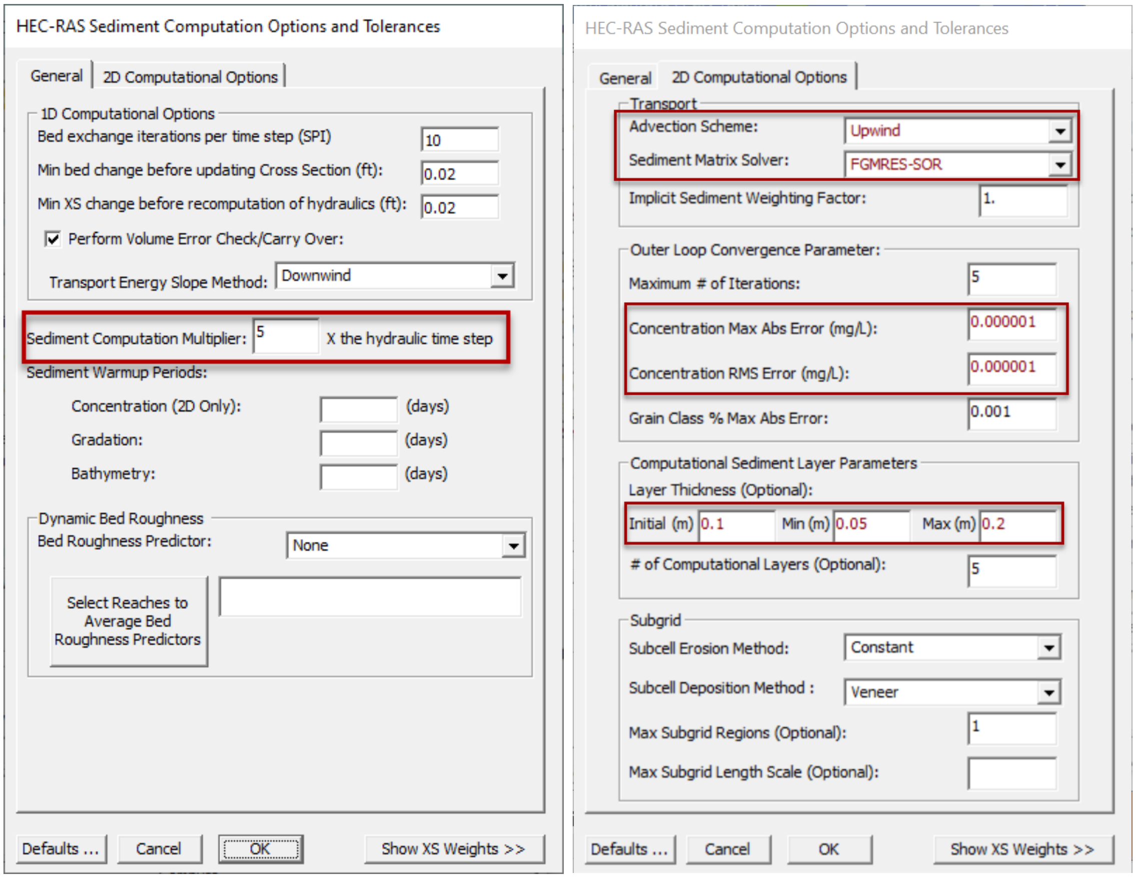
On the **General** tab, set the **Sediment Computation Multiplier** to **5X** the hydraulic time step.

On the **2D Computational Options** tab set the **Advection Scheme** to upwind.

We often change the **Sediment Matrix Solver** to speed models up, but we will leave it for this short model.

Most models leave the **Concentration Max Error and Concentration RSM Error** default, but because this is a flume model we will reduce these to **0.000001**.

Layer thickness sets your vertical computational intervals. The defaults are set for a river. So we need to reduce these for a flume to **Initial=0.05m**, **Min=0.005m**, and **Max=0.2m**.



3.5.11 Sediment Output Options

3.6 Run the Model

Go back to the Unsteady Flow Analysis window and click on the compute button. The model should start running and finish within a few minutes.

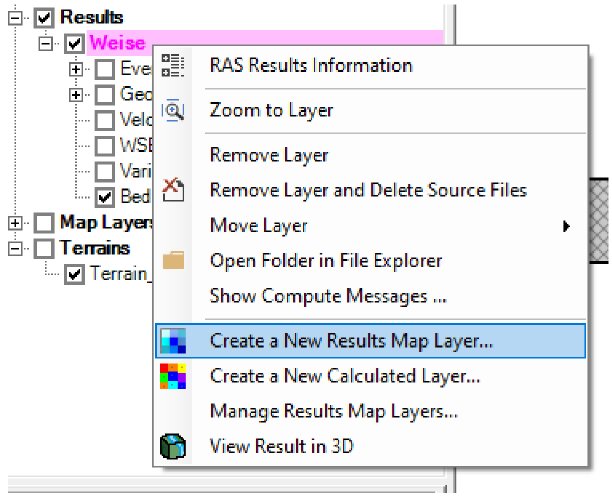
3.7 Open the Computation Log File

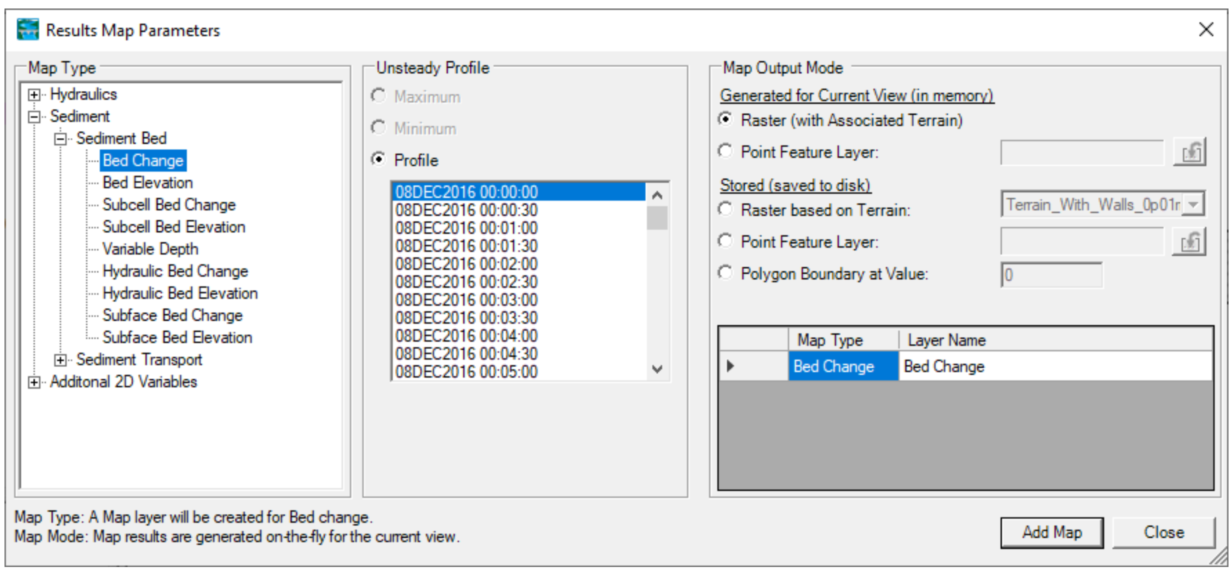
The computation log file contains detailed information on the model setup and variable statistics during the simulation. This file is very important to look at when running sediment. It contains detailed information on the sediment model setup, initial and boundary conditions which should be checked to make sure everything is correction.

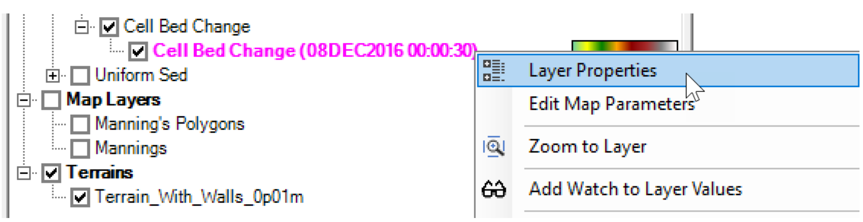
3.8 View Results in RAS Mapper

Open **RAS Mapper** from the main HEC-RAS menu by clicking on . Expand the **Results** node and select the 2D area you just ran. Turn on the computational mesh.

To add a 2D sediment output variable, right-click on the 2D area and select the menu **Add New RAS Results Map Layer**.

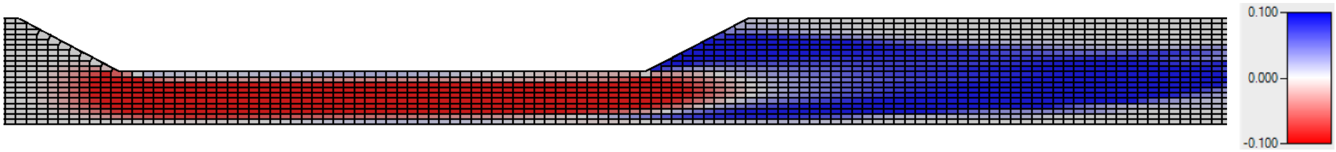






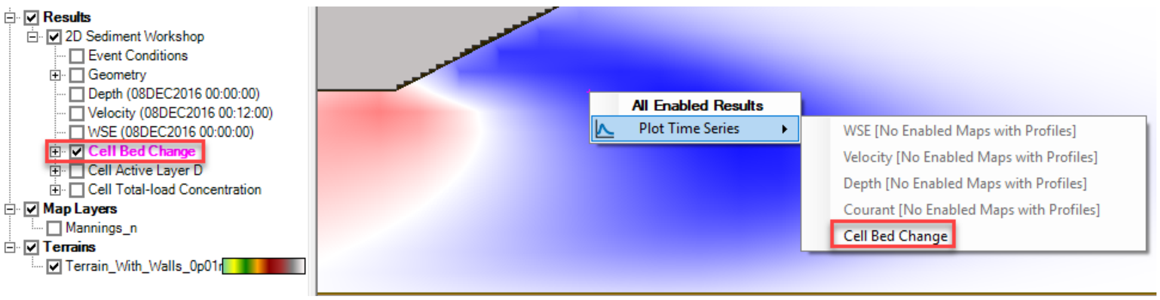
Adjust the **Color Ramp** and for the dataset so that it can be visualized well in **RAS Mapper**.

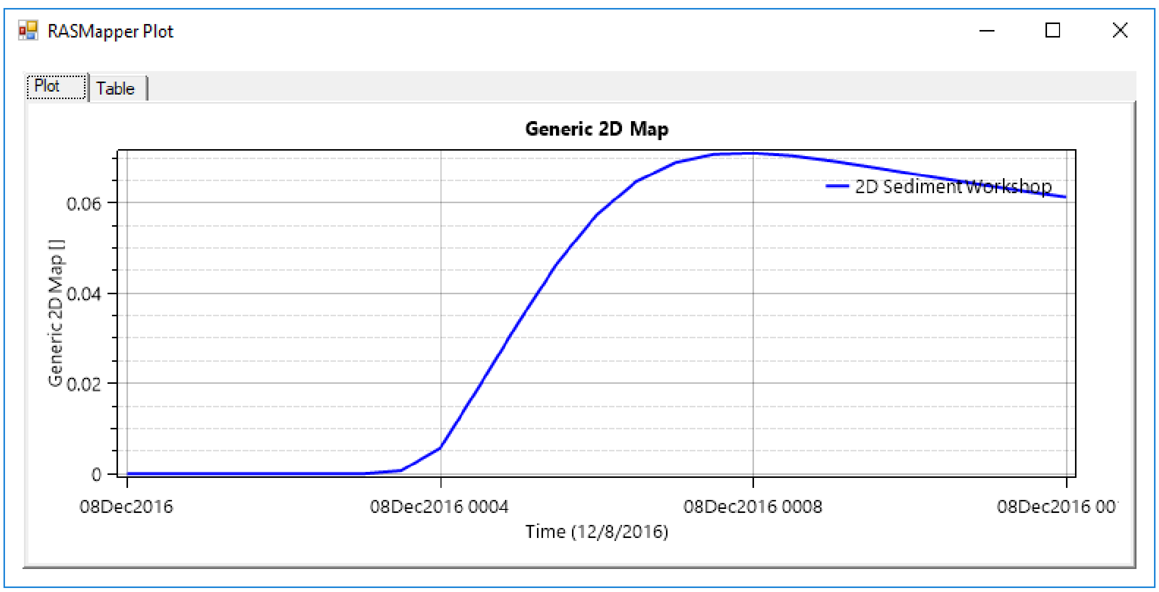
Below is an example of the computed bed change with a red-white-blue color ramp with lower and upper limits set to -0.1 m and 0.1 m, respectively.



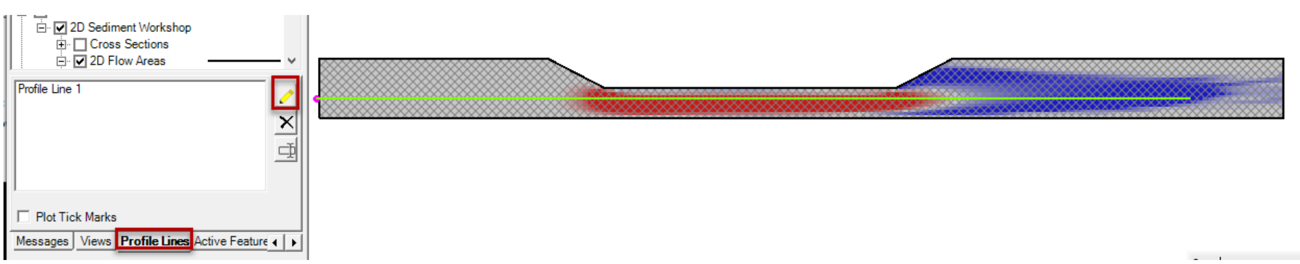
Try adding other 2D variables from both the **Sediment Bed** and **Sediment Transport Output Blocks**.

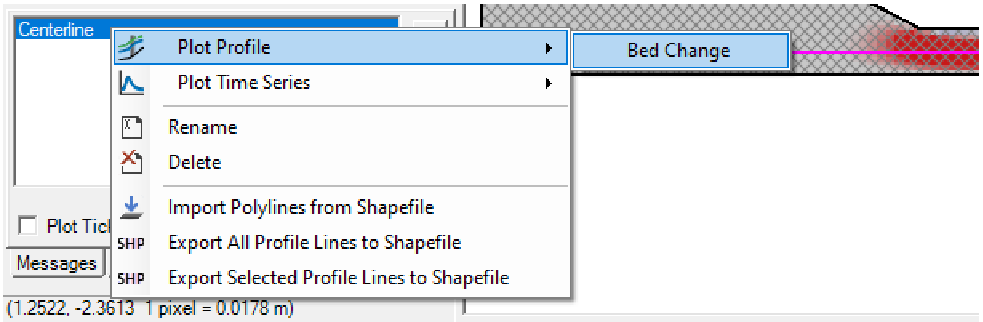
3.8.12 Plot Time Series

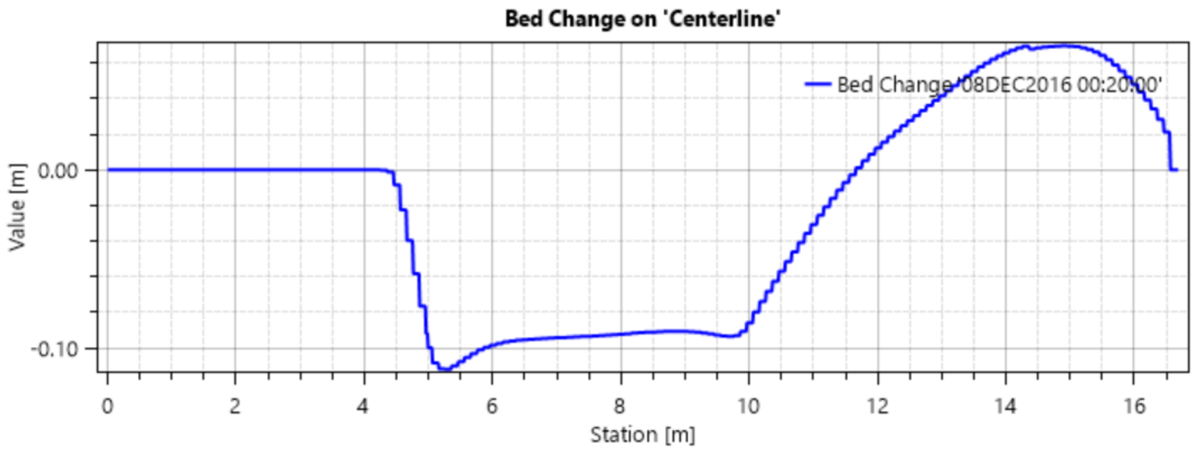




3.8.13 Plot Profile







Experiment with other results like Concentration.

Add “Depth” under the Hydraulics Results.
Compare it to the “Variable Depth” results.
How are they different?