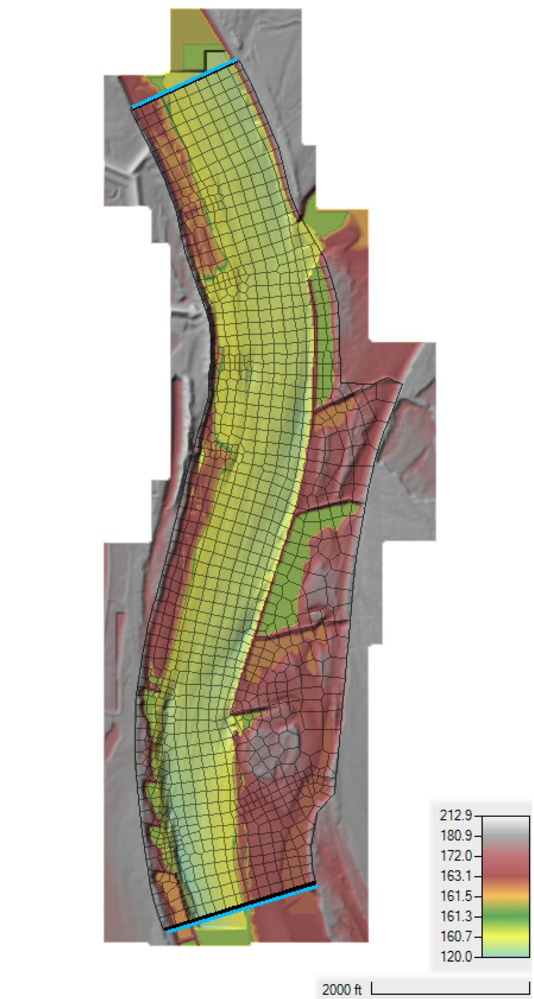
Initializing a 2D Sediment Model
Workshop

1 Objective

The goal of this workshop is to understand how to initialize a 2D sediment model. The workshop covers how to initialize a 2D hydraulics model, hydraulic and sediment warmup periods, initial condition points, hotstarts, and restart files.

2 Introduction

The model is based on a small reach of the Arkansas River. For the purposes of the workshop, the model is setup is relatively simple and fast. The model extent is relatively small model, and the mesh has a coarse resolution. The computational mesh, boundary condition lines, and terrain are shown in the figure below.



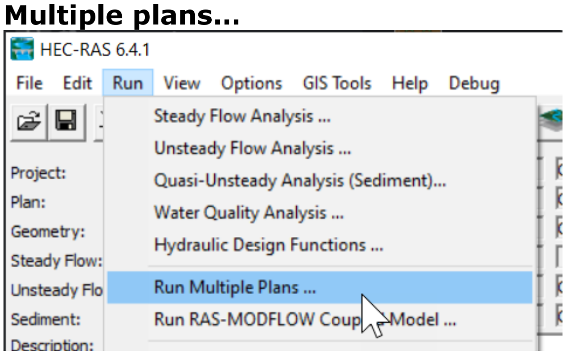
The base model has a constant flow upstream of 70,000 cfs, and a constant stage downstream of 172.936 ft. A morphologic acceleration factor of 10 is utilized to simulate 10 days of bed change over one day of simulation time. With this model setup, each run takes about 5-8 seconds to run which good for sensitivity analysis. It is important to note, that though the current model setup may not be good for final production runs of a project, the sensitivity analysis as applied here with a fast model setup is useful and recommended for projects.

3 Hydraulic Model Initialization

In this section, some basic approaches for initializing a hydraulic model are covered. These include warmup period, initial stages from boundary conditions, and initial stages from initial condition points.

3.1 No Warmup or Initial Conditions

1. **Start HEC-RAS**.
2. **Open** the HEC-RAS project names "ArkansasPool2.prj" in the "2.4 W - Initializing a 2D Sediment Model" workshop folder.
3. From the main HEC-RAS editor, click on the **Run** menu and select **Run Multiple plans…**



4. **Open** the plan "Flow Only - No Wamup IC".
5. **Run** the plan and inspect the results in RAS Mapper.

Question: The model shows instabilities at the beginning of the simulation. Explain what is causing this issue.

3.2 Initial BC Stage

One simple way of initializing the stage in a 2D model is to utilize the stage information from boundary conditions. HEC-RAS has the option to flood the 2D domain using the 2D stage boundary conditions.

6. **Open** the **Unsteady Flow Data** editor.
7. **Save As** the unsteady flow file as "Stage IC".
8. **Open** the stage hydrograph boundary condition.
9. **Check** the box "Use Initial Stage".
10. **Save** the plan with the **Title** and **Short ID** "Flow Only – No Warmup - Stage IC".
11. **Run** the plan.
12. **Inspect** the water levels and time series by plotting time series near the downstream boundary.

Question: How long does it take for the model to reach steady state conditions?

3.3 Warmup with Initial BC Stage

The estimate of how long it takes for the model to reach a steady stage condition is a good reference for what to use for the warmup period. The warmup period ramps up the boundary conditions during a portion of the warmup period and then holds the value constant for the remaining portion.

13. **Save As** the plan with the **Title** and **Short ID** "Flow Only - Wamup - Stage IC".
14. **Click** on the menu **Options | Computational Options and Tolerances** in the **Unsteady Flow Analysis** editor.
15. **Click** on the **2D Flow Options** tab.
16. **Set** the **Initial Conditions Time** to 2 hrs.
17. **Run** the plan.
18. **Inspect** the water levels and time series by plotting time series near the downstream boundary.

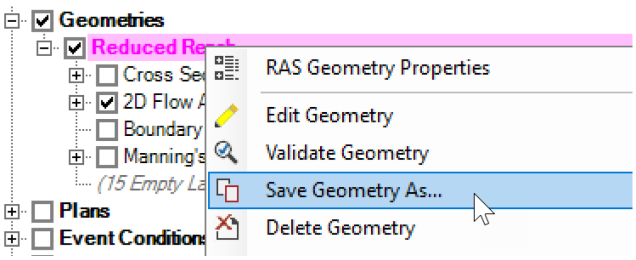
Question: How did the results change?

Question: Are there any initial conditions effects still present at the beginning of the simulation?

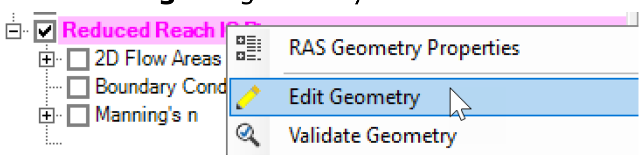
3.4 Warmup with Initial Condition Points Stage

For large and/or more complex models it is often useful to specify Initial Condition Points with stages to initialize the model. The provides the model with an initial sloping water surface which can lead to shorter warmup periods.

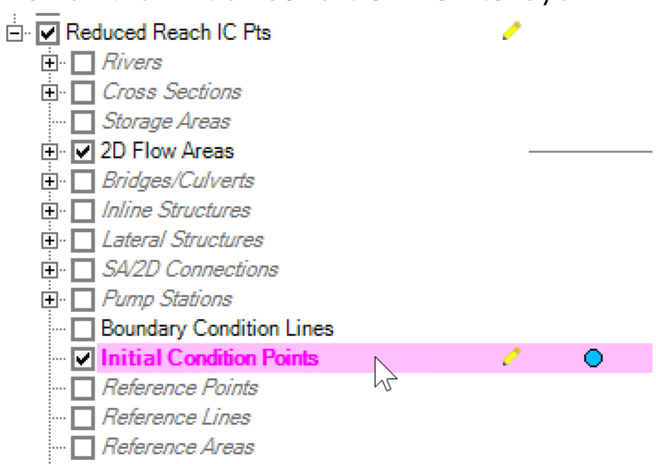
19. **Open RAS Mapper.**
20. **Save** a copy of the geometry as "Reduced Reach IC Pts".



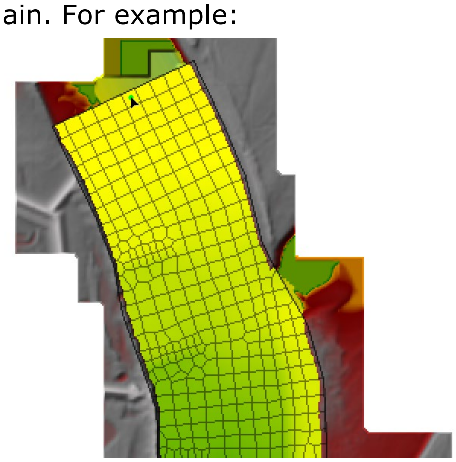
21. Start **Editing** the geometry.



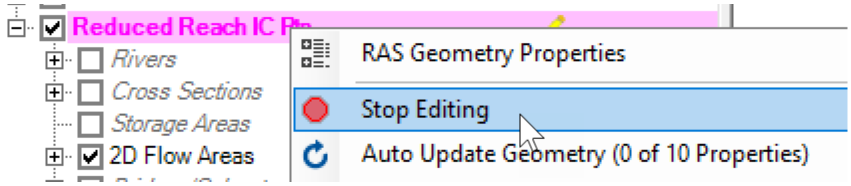
22. **Click** on the **Initial Condition Points** layer.



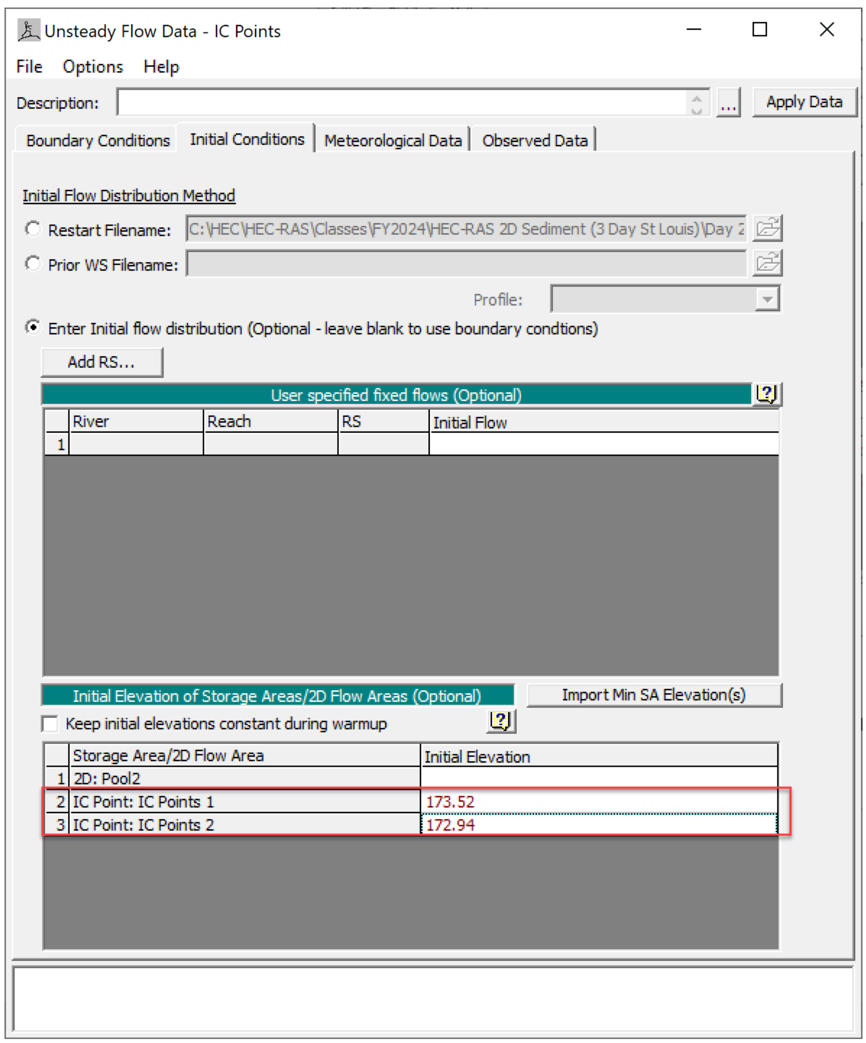
23. **Add** Initial Condition Points near the upstream and downstream ends of the domain. For example:



24. **Name** the point and click **OK**.
25. **Stop Editing** the geometry by right-clicking on the geometry layer and selecting **Stop Editing**, and click **Yes** to save the changes.



26. **Open** the **Unsteady Flow Data** editor.
27. **Save As** the unsteady flow file as "IC Points"
28. **Set** the plan Geometry to "Reduced Reach IC Pts" in the **Unsteady Flow Analysis** editor.
29. **Save As** the plan with **Title** and **Short ID** "Flow Only – No Wamup - IC Pts".
30. **Open** the **Unsteady Flow Data** editor.
31. **Click** on the **Initial Conditions** tab.
32. **Enter** an initial water surface at the initial condition points as shown below.



33. **Save** the Unsteady Flow Data and **Close** the editor.
34. **Set** the **Warmup** period to zero. The purpose of this is to see the effect of the initial water surface on the model initial condition.
35. **Run** the plan.
36. **Compare** the results.

Question: How could the initial conditions be improved?

Question: Would a Diffusion Wave model have a longer or shorter initialization period and why?

4 Sediment Model Initialization

In this section, different approaches and strategies are covered for initializing 2D sediment models. Note that even thought the simulations are setup with a 1-day simulation window, the bed change actually represents 10 days of bed change because the morphological acceleration factor is set to 10.

4.5 Sediment Warmup Periods

The simplest way to add initialization options to a sediment model, especially if there are no prior simulation results which can be used to initialize the sediment model, is to setup **Sediment Warmup Periods**. The sediment warmup periods are run after the hydraulic **Initial Conditions Period**.

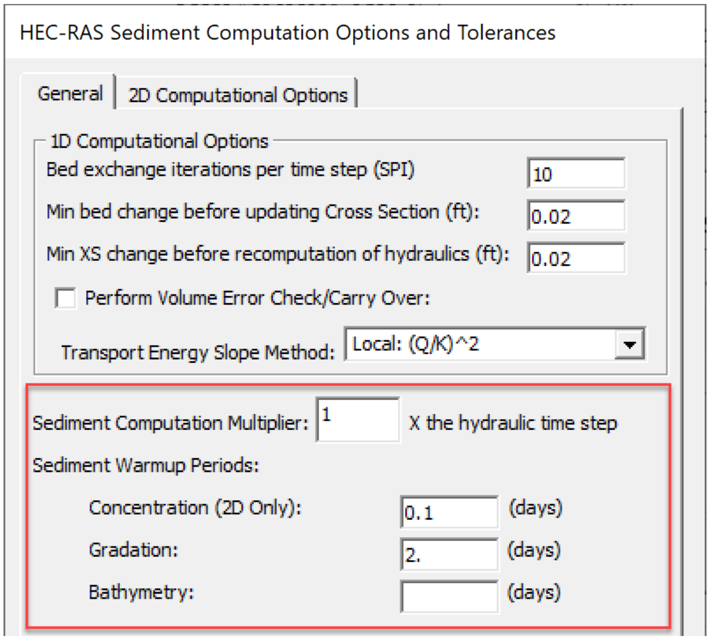
37. **Open** the base sediment plan titled "Sed Base". This plan has no special treatment for sediment initial conditions.
38. **Run** the plan.
39. **Open RAS Mapper** and inspect the results by plotting the **Bed Change**, **Active Layer Percentile Diameters**, **Sediment Concentrations**, and **Sediment Concentration Capacities**.
40. **Plot** time series of each of the variables at different model locations.

    Question: What do you notice about the sediment variable time series in terms of their variability at the beginning and end of the simulation?

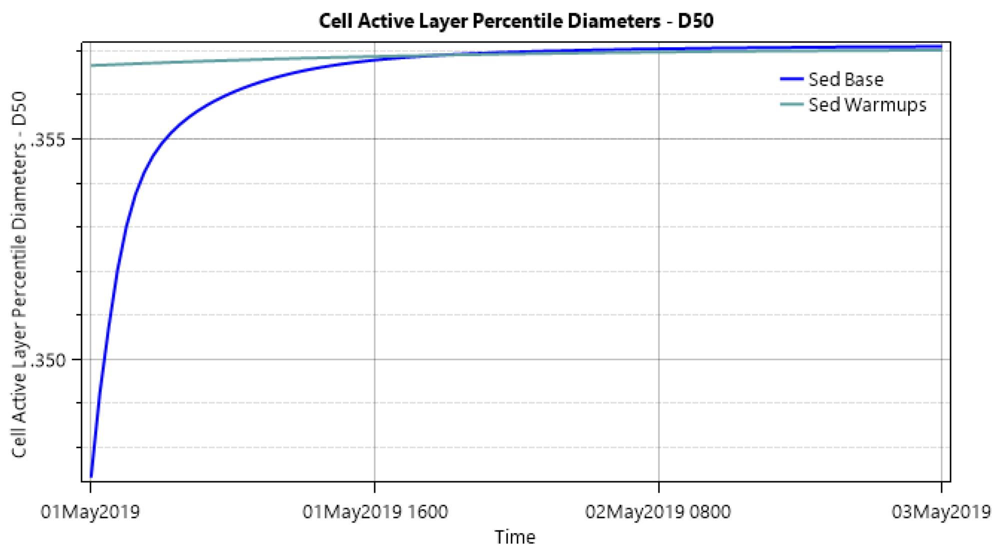
    Question: Do all the variables behave the same and can explain why?

41. **Save As** the plan with the **Title** and **Short ID** "Sed Warmups".
42. **Set** the **Sediment Warmup Periods** as shown below.

    Note: HEC-RAS 6.4.1 has a bug where the gradation warmup period does not work when utilized in combination with a **Sediment Computation Multiplier** larger than 1.0. Therefore, it will also be necessary for the purposes of this workshop to set the **Sediment Computation Multiplier** to 1.0, as shown below.



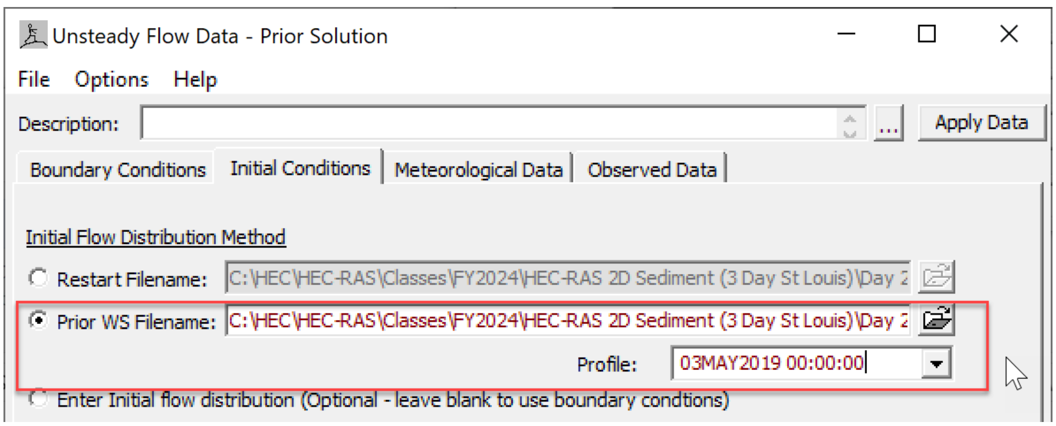
43. **Save** and **Run** the plan.
44. **Add** the **Active Layer Percentile Diameters** result to **RAS Mapper**.
45. **Compare** the initial median grain size for the two sediment plans by plotting the spatial maps and time series at a few points. Below is an example time series.



4.6 Prior Solution

Another way to initialize a sediment model is utilize a prior simulation result to initialize sediment. This approach has the advantage of not requiring outputting an Initial Conditions or Restart file (*.rst). The approach reads as much information as possible from a previous simulation to initialize sediment but since the result file typically does not contain all the state variable information (i.e. mass fractions and layer thickness for every bed layer). This option needs to be done in two steps. First the result file is specified for hydraulics. The hydraulics model will read and initialize water surfaces and current velocities from the initial conditions file. In the second part, the result file is specified again but for sediment transport. Since in general, not all of the bed information is written out to the result file, the model will try to initialize the model based on the information available in the file.

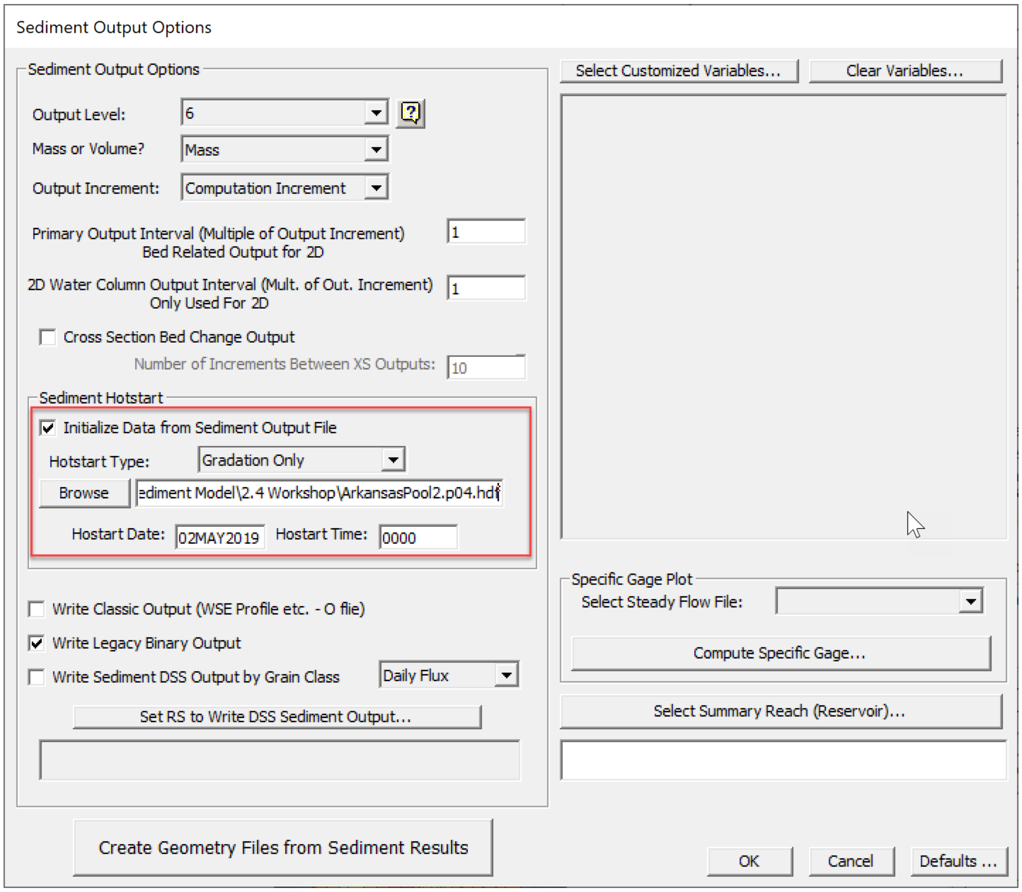
46. **Open** the **Unsteady Flow Data** editor.
47. **Open** the file "Clip Warmup 1hrIC".
48. **Save As** the file as "Prior Solution".
49. **Click** on the **Initial Conditions** tab.
50. **Check** the box labeled "Prior WS Filename" and browse to the plan which represents the "Sed Base" plan. Then select the last output time under in the Profile drop down. See image below



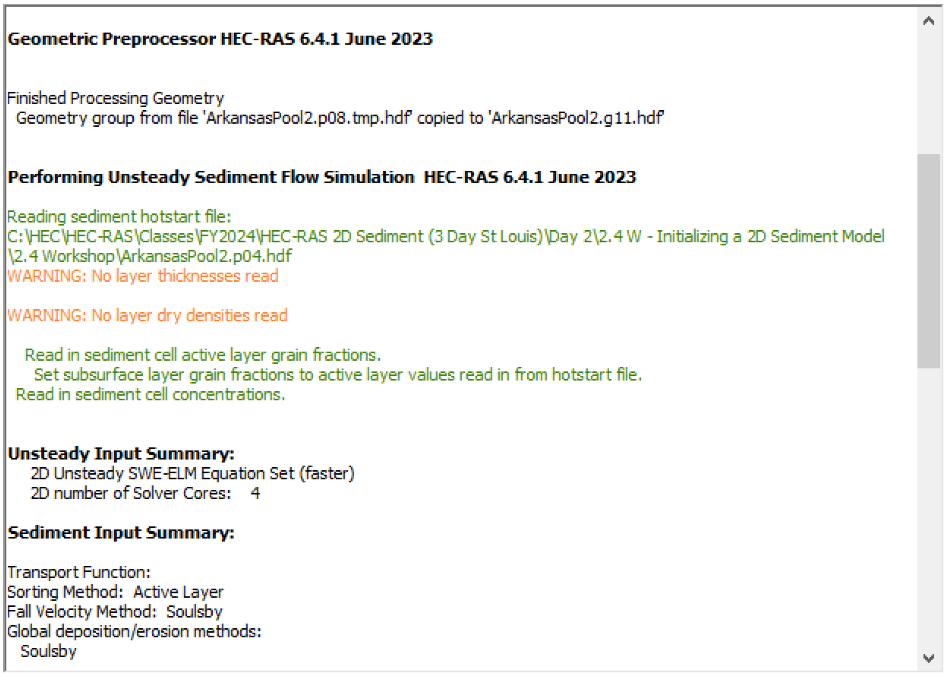
51. **Save** the **Unsteady Flow Data**.
52. **Open** the **Unsteady Flow Analysis** editor.
53. **Save** the plan with **Title** and **Short ID** "Sed Prior Solution".
54. **Set** the **Sediment Warmup Periods** to zero or delete them from the **Sediment Computation Options and Tolerances** editor.
55. **Set** the hydraulic **Initial Conditions Period** to zero or delete it from the hydraulic **Computational Options and Tolerances** editor.
56. **Open** the **Sediment Output Options** editor.

57. **Set** the **Sediment Hotstart** options as shown in the figure below. **Check** the box labeled "Initialize Data from Sediment Output File". Then set the **HotStart Type** to **Gradation Only**. Then set the **Hotstart Date** and **Hotstart Time** to **02MAY2019** and **0000**, respectively.

    Note: The plan should correspond to the same plan utilized above as the **Prior WS Filename**.



58. **Run** the plan. The runtime messages should look something like:

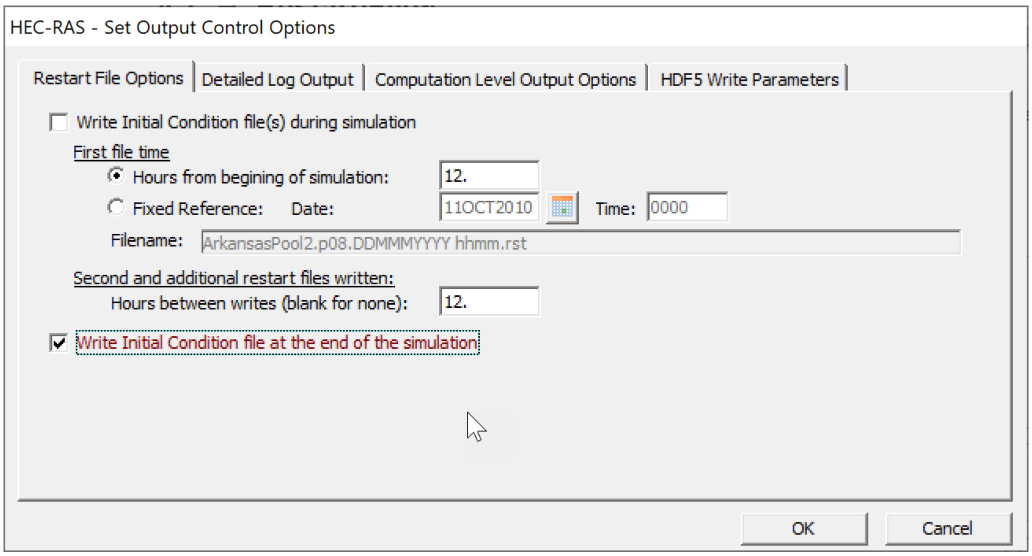


59. **Compare** the results with the previous sediment simulations and confirm that the sediment model was initialized using the previous simulation results.

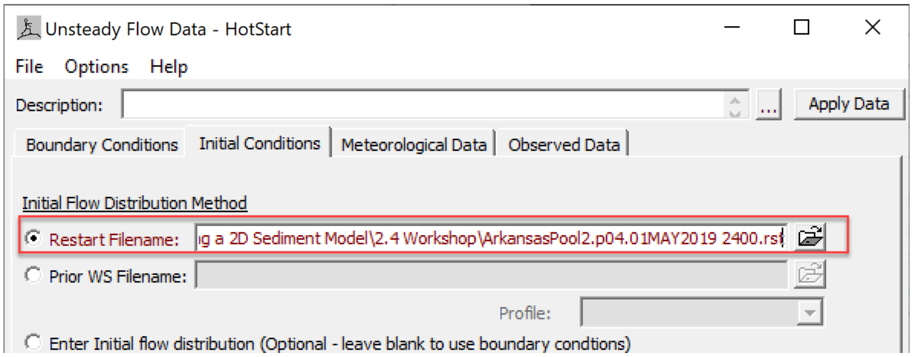
4.7 HotStart

Another way to initialize a sediment model is utilize a restart file. The restart file has the advantage of containing all the state variables. The disadvantage of the Restart file is that it does not allow the user to select which variables are used. When utilizing a restart file, all of the variables used.

60. **Open** the **Unsteady Flow Data** editor.
61. **Open** the plan **Titled** "Sed Base".
62. **Save As** the plan with the **Title** and **Short ID** "Sed Restart".
63. **Click** the menu **Options | Output Options**.
64. **Click** on the **Restart File Options** tab.
65. **Check** the box labeled "Write Initial Condition file at the end of the simulation".



66. **Save** the plan.
67. **Run** the plan. You'll notice that the project folder will contain a file with a name like "ArkansasPool2.p04.01MAY2019 2400.rst". The file name contains the restart time. When the restart is utilized to initialize another plan, the model initial time does not have to match the restart time.
68. **Open** the **Unsteady Flow Data** editor.
69. **Save As** the unsteady flow file with the name "HotStart".
70. **Click** on the **Initial Conditions** tab.
71. **Check** the box **Restart Filename** as shown below.

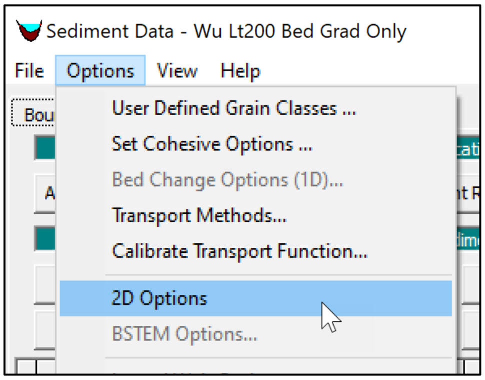


72. **Save** the unsteady flow data and close the editor.
73. **Open** the **Unsteady Flow Analysis** editor.
74. **Save** the plan with the **Title** and **Short ID** "Sed HotStart".
75. **Delete** the **Initial Conditions Period** in the **2D Flow Options** tab of the **Computation Options and Tolerances** editor. Since the model is being hotstarted, this parameter is no longer needed.
76. **Run** the plan.

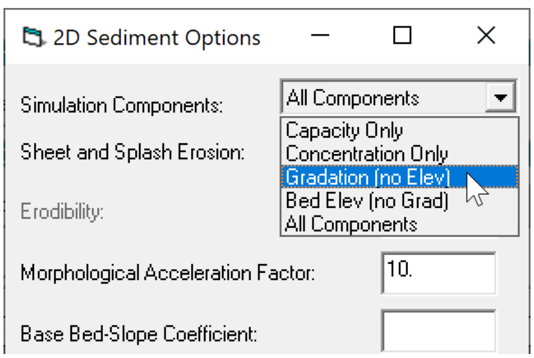
4.8 Simulation Components and HotStart

One of the issues with using a hotstart file for sediment is that it's not possible to control what variables are read-in and initialized. A simple way around this issue is to utilize the **Sediment Simulation Components**. In the example below, a simulation is running with only bed gradations and not updating bed elevations. A restart file is written from the end of this simulation and used to hotstart a subsequent simulation.

1. **Open** the **Unsteady Flow Data** editor.
2. **Open** the plan **Titled** "Sed Base".
3. **Open** the **Sediment Data** editor.
4. **Save As** the sediment data as "Wu Lt200 Grad".
5. **Click** on **Options | 2D Options**.



6. **Set** the **Simulation Components** drop down menu to **Gradation (no Elev)**.

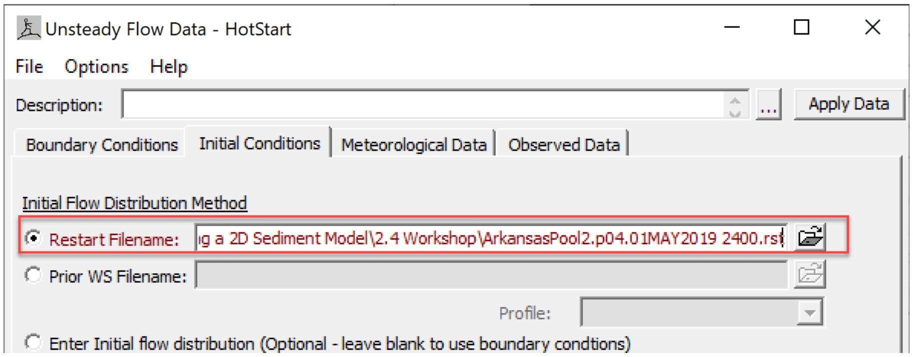


7. **Click OK**.
8. **Save** and **close** the **Sediment Data** editor.
9. **Open** the **Unsteady Flow Data** editor.
10. **Save As** the plan with the **Title** and **Short ID** "Sed Restart Grad".
11. **Run** the plan and inspect the results.

Question: Did the model compute any bed change?

Question: Does this model conserve mass locally and globally?

12. **Open** the **Unsteady Flow Data** editor.
13. **Save As** the unsteady flow file with the name "HotStart Grad".
14. **Click** on the **Initial Conditions** tab.
15. **Check** the box **Restart Filename** as shown below.



16. **Save** the unsteady flow data and close the editor.
17. **Open** the **Unsteady Flow Analysis** editor.
18. **Open** the plan titled "Sed HotStart".
19. **Set** the **Unsteady Flow File** to "Hotstart Grad".
20. **Save** the plan with the **Title** and **Short ID** "Sed HotStart Grad".
21. **Run** the plan and inspect the results.