
Creating a Simple 2D Model with HEC-RAS

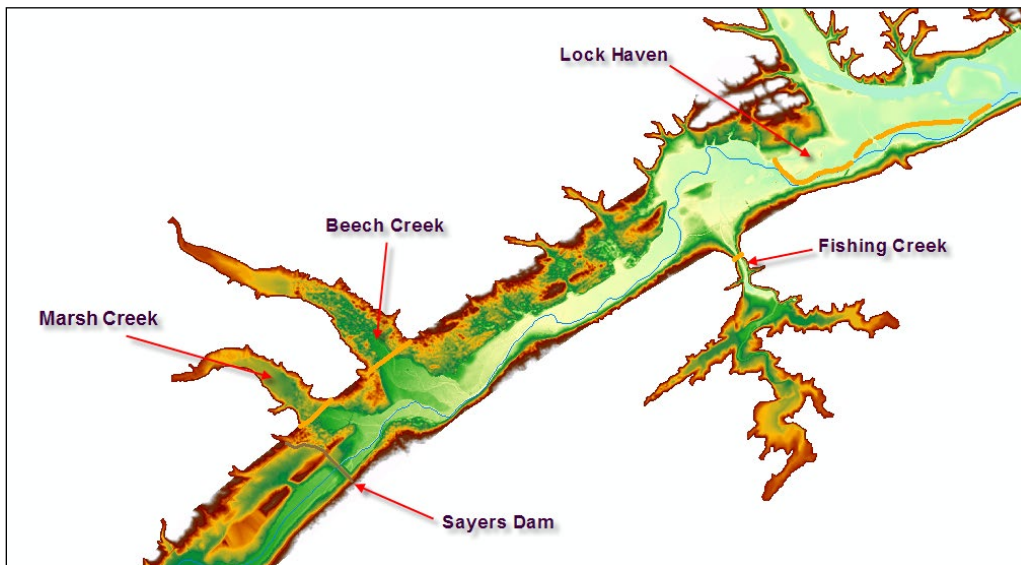
Workshop

1 Objective

In this workshop, you will learn how to create a simple 2D model with HEC-RAS of the main channel and floodplain area. This workshop will require you to use multiple terrain datasets to form one terrain model, develop a 2D mesh, provide inflow hydrograph information, downstream boundary conditions, and perform model runs.


2 Background

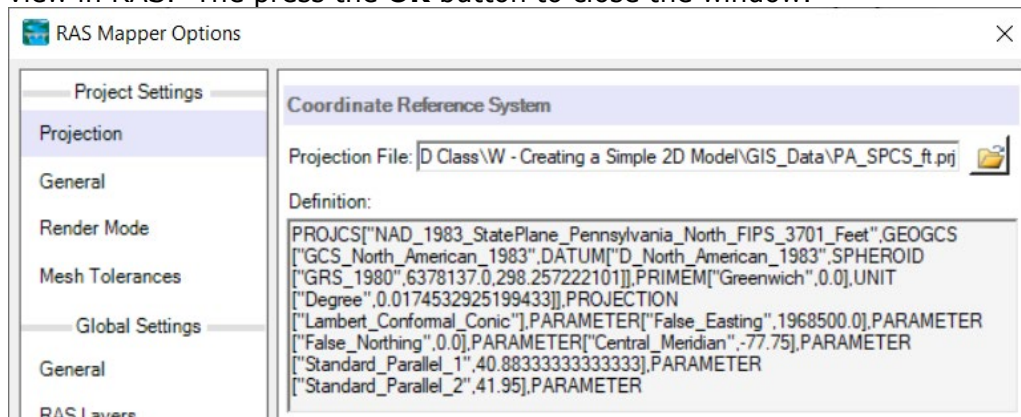
You will be working on Bald Eagle Creek near Lock Haven, PA. Flow hydrographs will be used to simulate flows from Joseph Sayer's Dam downstream to the town of Lock Haven and the confluence with the West Branch Susquehanna River. Inflows will also be specified for local tributaries on Marsh Creek, Beech Creek, and Fishing Creek. A Normal Depth Boundary Condition will be used.




3 Terrain Model Preparation

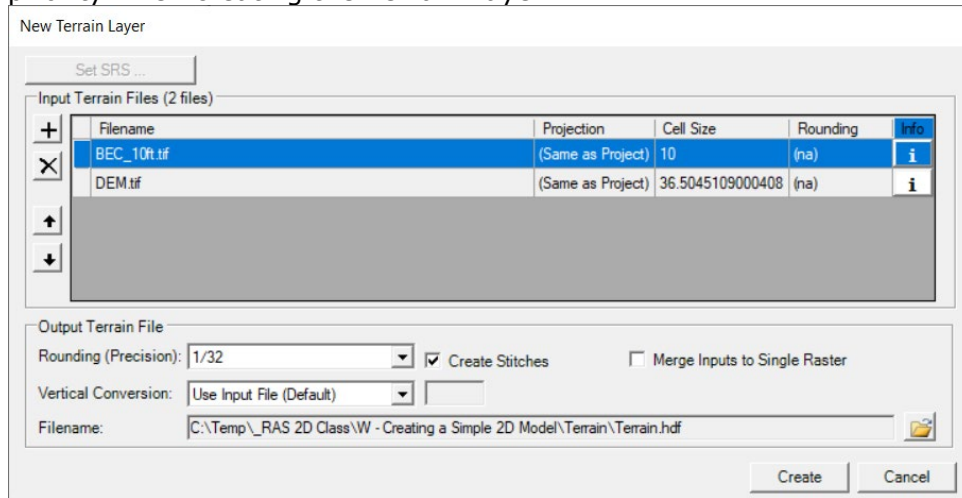
This part of the workshop will guide you through the process of importing terrain data. The terrain data will be used as the basis for the mesh used for 2D hydraulic computations.

1. **Start HEC-RAS.**
2. Start a NEW project using **File | New Project...** Go to the workshop directory for this workshop ("Creating a Simple 2D Model") and then providing a **Title** and **File Name**. Press **OK** to save it.
3. Launch **RAS Mapper** .
4. Select **Project | Set Projection**. Select the Folder button at the top left and navigate to the "**PA_SPCS_ft.prj**" provided in the "GIS_Data" folder. Select and Open this file. This sets the coordinate system for all the data you will view in RAS. The press the **OK** button to close the window.

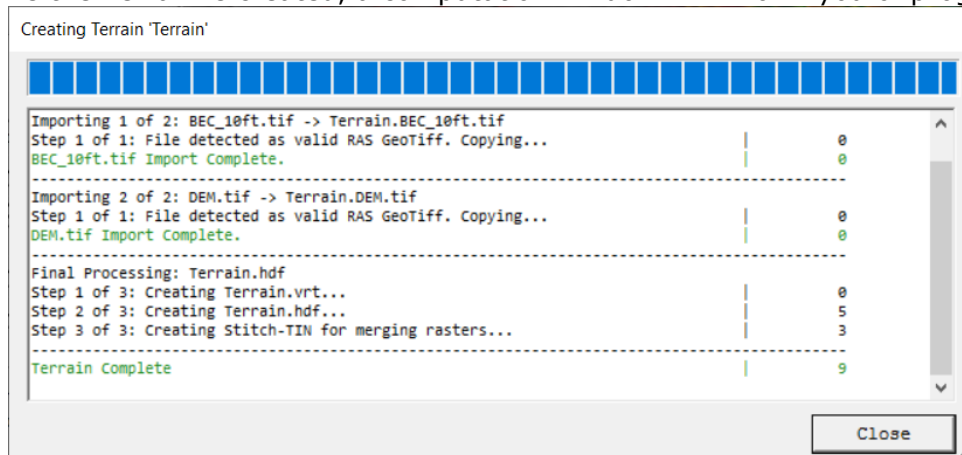


5. Import terrain data for use in RAS by selecting the **Project | Create New RAS Terrain** menu item.
6. Click the **Add Files**  button and navigate to the "Terrain" folder. Select the "**BEC_10ft.tif**" and "**DEM.tif**" files, then press the **Open** button at the bottom. The "BEC_10ft.tif" file contains gridded data at 10ft postings for the channel created from a TIN and the "DEM.tif" is a DEM (lower resolution) downloaded from the USGS for the remainder of the floodplain.

- Use the New Terrain Layer dialog to order the files for import. Note that the “most important” layer should be on the top of the list and will get the highest priority when creating the Terrain Layer.

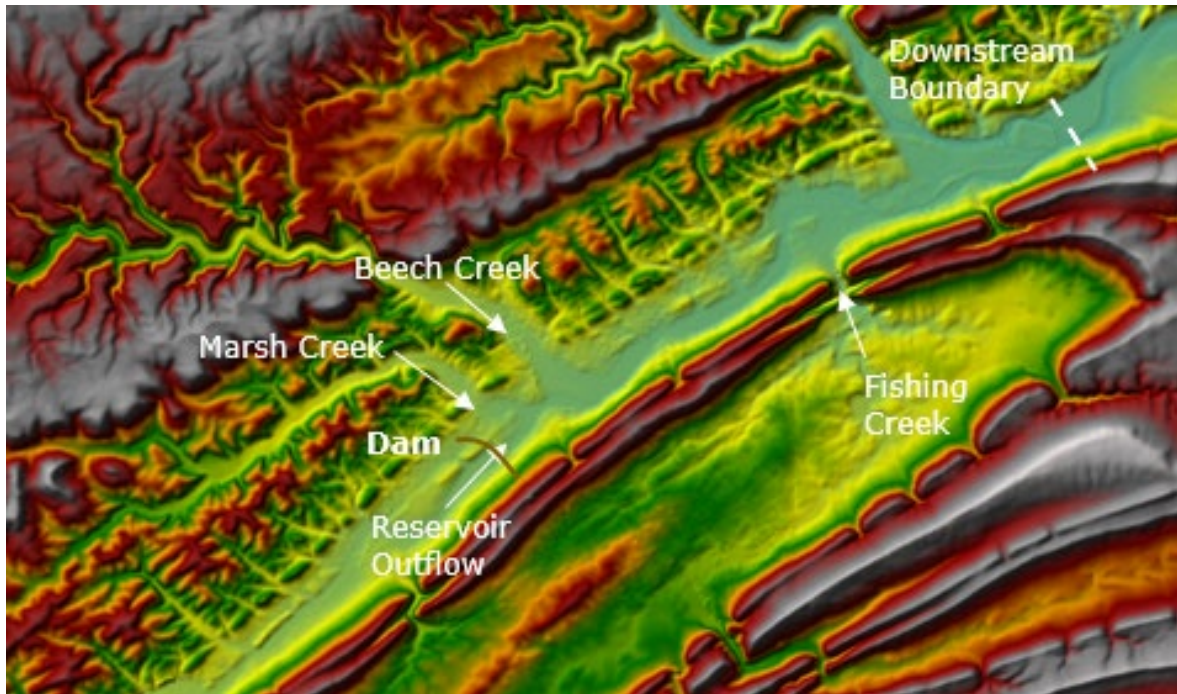


- Press the **Create** button. As the Terrain is created, a computation window will inform you of progress.



- When the terrain process is finished, select **Close** on the processing window, this will close both windows. Turn on the Terrain Layer.
- Right-click** on the Terrain Layer and choose **Zoom to Layer** (if necessary to see the terrain).
- Double-click** on the **Terrain** Layer to access its Properties.


Below is what the terrain should look like. Also shown are all the boundary condition locations for this workshop.



Note the location of Sayer's Dam and tributaries, shown in the background image above. You will be creating a 2D Model **below** the Dam with tributary inflows.

4 Create the 2D Flow Area

The 2D Flow Area will go from just **downstream of the Dam** to the confluence with West Branch Susquehanna River and will include a portion of Marsh Creek, Beech Creek, and Fishing Creek tributaries.

1. Right-click on the **Geometries** group and choose **Add New Geometry**. Enter "Simple Geometry" for the name. Press **OK** to create the geometry layer.
2. Right-click the new Geometry Layer and select **Edit Geometry**.
3. Select the 2D Flow Areas **Perimeter layer**. Draw the perimeter using the **Add New Feature**  tool.

Question: What things should you consider when creating/defining the 2D Flow Area polygon?

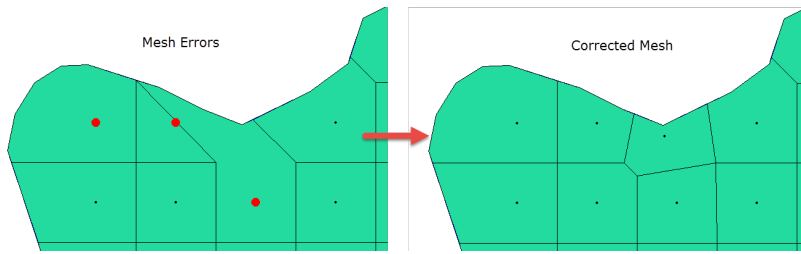
4. Enter a name, when finished.

A dialog box titled "Name Perimeter" with a text input field containing "2D Flow Area". Below the input field are "OK" and "Cancel" buttons. The text "Provide a unique name for the Perimeter" is displayed above the input field.

5. In the **Edit 2D Area Properties** that come up, Enter **500** for the **DX and DY Point Spacing**.
6. Click on the **Generate Computation Points with All Breaklines** button.
7. Change the **Default Manning's n Value** to **0.04**.

The "2D Flow Area Editor" dialog box is shown. The "2D Flow Area" dropdown menu is set to "2D Flow Area". Under "Cell Properties", the "Computation Points" section has "Points Spacing (ft) DX" and "DY" both set to "500". The "Generate Computation Points with All Breaklines" button is highlighted. The "Hydraulic Cell/Face Properties" section has "Default Manning's n Value" set to "0.04". The "Compute Property Tables" button is visible. The "Mesh State" is "UnknownError" and "Mesh Status" is "Mesh not created".

8. **Close** the Editor.
9. Turn on the **Computation Points**.
10. **Inspect the edge of the 2D Flow Area Mesh** for any mistakes in the boundary. Make sure each cell has only one point in it. Turn off all background layers to more easily inspect the mesh. (Mesh problems should show up with cell points colored red.)
To fix errors in the mesh, use the Editing Tools to Add, Delete, or Move points.



11. Stop Editing to **Save** the Geometry!

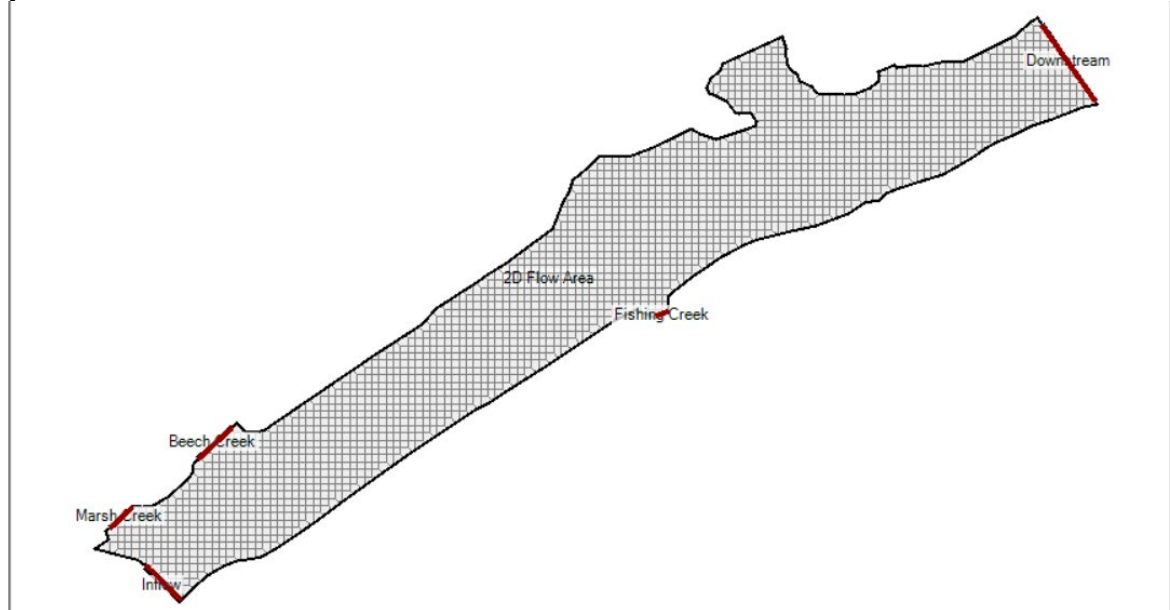
5 Flow Data Connections

After creating the 2D Flow Area computation mesh, add all of the locations for flow boundary conditions.

5.1 RAS Mapper

12. **Start Editing** in RAS Mapper.
13. Select the **BC Lines** layer.
14. Create boundary conditions, left to right when looking downstream at each of the boundary locations shown below. Enter a name for each (Inflow, Beech Creek, Marsh Creek, Fishing Creek and Downstream).

*****WARNING: Boundary Conditions must be drawn outside of the perimeter of the mesh!*****



5.2 Geometric Editor

15. Close RAS Mapper.
16. Open the Geometric Schematic and **Open** the new **Geometry** ("Simple Geometry").

Open Geometry File


Selected File Title	Filename
Simple Geometry	C:\Temp_RAS 2D Class\W - Creating a Simple 2D Model\Simple2DModel.g01
Simple Geometry	C:\Temp_RAS 2D Class\W - Creating a Simple 2D Model\Simple2DModel.g01

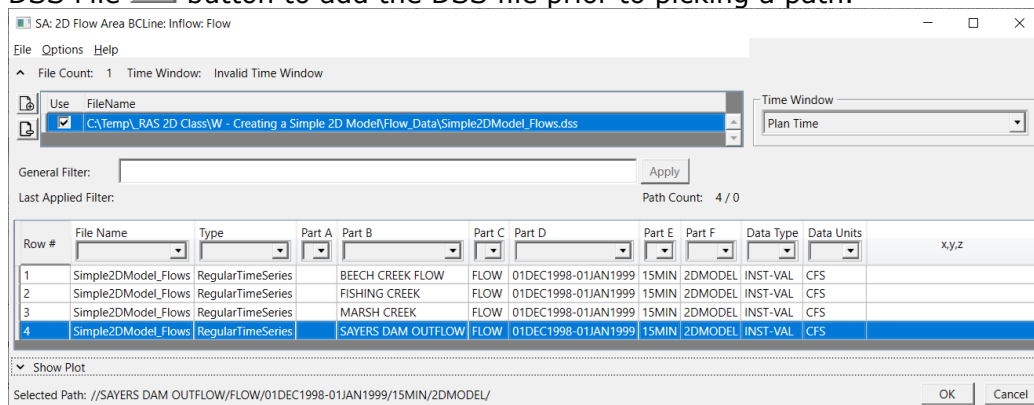
17. Verify the Geometry came in correctly. If not, fix it.
18. Close the Geometric Editor.

5.3 Unsteady Flow Editor

19. Open the **Unsteady Flow Data** editor and enter boundary condition information.

	River	Reach	RS	Boundary Condition
Storage/2D Flow Areas				
1	2DArea	BCLine: Inflow		Flow Hydrograph
2	2DArea	BCLine: Marsh Creek		Flow Hydrograph
3	2DArea	BCLine: Beech Creek		Flow Hydrograph
4	2DArea	BCLine: Fishing Creek		Flow Hydrograph
5	2DArea	BCLine: Downstream		Normal Depth

20. Use **Normal Depth** with Slope = **0.001** for the downstream boundary condition.
21. Use the "**Simple2DModel_Flows.dss**" file to define the **Flow Hydrograph** for the upstream boundary and tributaries. You will need to click the Add DSS File  button to add the DSS file prior to picking a path.



Row #	File Name	Type	Part A	Part B	Part C	Part D	Part E	Part F	Data Type	Data Units	x,y,z
1	Simple2DModel_Flows	RegularTimeSeries		BEECH CREEK FLOW	FLOW	01DEC1998-01JAN1999	15MIN	2DMODEL	INST-VAL	CFS	
2	Simple2DModel_Flows	RegularTimeSeries		FISHING CREEK	FLOW	01DEC1998-01JAN1999	15MIN	2DMODEL	INST-VAL	CFS	
3	Simple2DModel_Flows	RegularTimeSeries		MARSH CREEK	FLOW	01DEC1998-01JAN1999	15MIN	2DMODEL	INST-VAL	CFS	
4	Simple2DModel_Flows	RegularTimeSeries		SAYERS DAM OUTFLOW	FLOW	01DEC1998-01JAN1999	15MIN	2DMODEL	INST-VAL	CFS	

Selected Path: //SAYERS DAM OUTFLOW/FLOW/01DEC1998-01JAN1999/15MIN/2DMODEL/

The hydrograph for the inflow is named "Sayers Dam Outflow". The other

DSS records are named the same as the boundary condition.

Flow Hydrograph

SA: 2D Flow Area BCLine: Inflow

Read from DSS before simulation

File: C:\Temp_RAS 2D Class\W - Creating a Simple 2D Model\Flow_Data\Simple2DModel_Flows.dss

Path: //SAYERS DAM OUTFLOW/FLOW/01DEC1998-01JAN1999/15MIN/2DMODEL/

22. Specify the "EG Slope for distributing flow along BC line" as **0.001**. (This is located at the bottom right on the Flow Hydrograph window.) This must be done for all of the flow hydrographs attached to the 2D Flow Area.
23. **Repeat** previous 2 steps for all **inflow** locations.
24. Save the Unsteady Flow data when complete (call it "**Flows**").

6 Plan and Simulation

25. Open the Unsteady Flow Analysis window, **Save a Plan**, and enter all of the necessary information to make a run. Set up the time window:
Start Date: 02JAN1999 Start Time: 0000
End Date: 06JAN1999 End Time: 0000
26. Turn on the Geometry Preprocessor and Unsteady Flow Simulation programs.
27. Specify a Time step.

Question: What is an appropriate time step to use for the simulation? How would you figure it out?

28. Save a plan by the name of the grid cell size and time step selected (“Initial Run”).
29. Run the simulation (press the **Compute** button).

Question: What SHOULD we have done before simulating?

30. Go to the **Options | Calculation Options and Tolerances** and set the **2D Flow Option** for the **Initial Conditions Time** (and Ramp Up Fraction).

Question: What is an appropriate Initial Conditions Time to use? What happens if you don’t use an Initial Conditions Time? How else could you set the initial conditions?