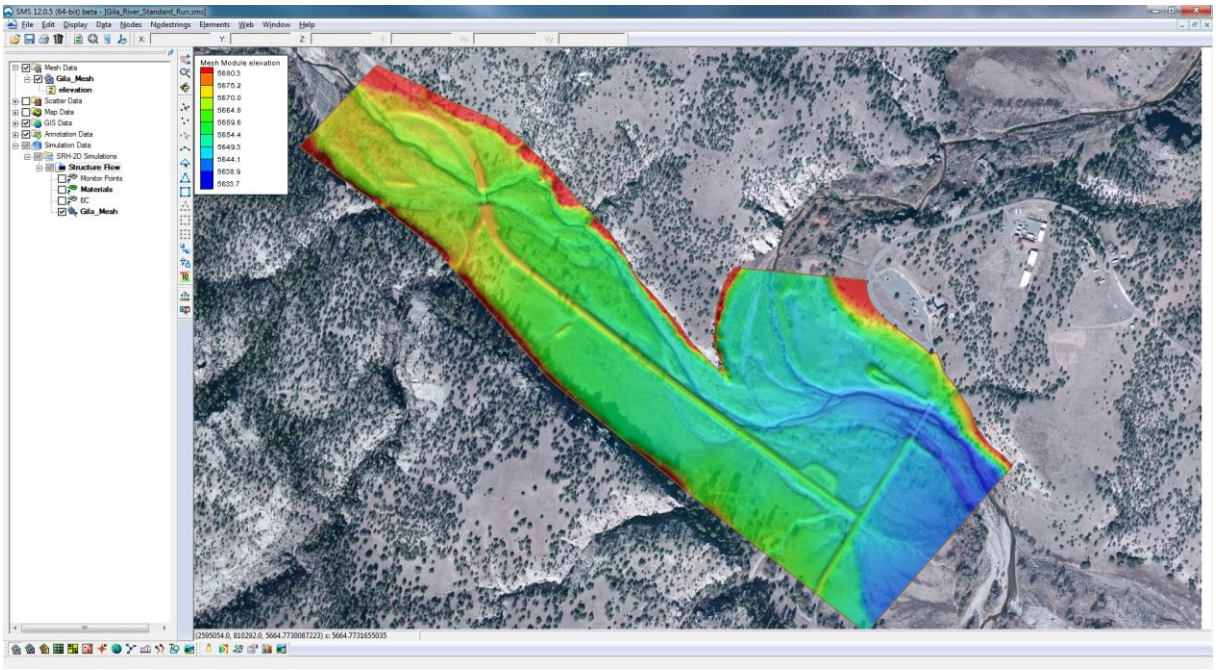
SRH-2D Tutorial

Simulations



Objectives

This tutorial will demonstrate the process of creating a new SRH-2D simulation from an existing simulation. This workflow is very useful when adding new features or complexity to a model.

Prerequisites

- SRH-2D Tutorial

Requirements

- SRH-2D
- Mesh Module
- Scatter Module
- Map Module

Time

- 15–20 minutes

1 Opening an Existing Project

To begin, an existing project will be loaded into SMS. In this project, an SRH-2D model has been created near a confluence along the Gila River, located in New Mexico. The model has been created to assess the effects different hydraulic structures will have on the Gila River.

This tutorial will demonstrate the process of duplicating an SRH-2D so that modifications such as changing the material type, adding a structure, or changing the inflow/outflow boundary conditions can be made and compared to the original simulation. Subsequent tutorials have been written to demonstrate the creation of hydraulic structures within an SRH-2D model and are available for download.

1. Open a new instance of SMS.
2. Select *File* | **Open**. This will bring up the *Open* dialog.
3. Select the file "Gila_Base.sms" from the "Data Files" folder in the "SRH2D_Simulation" folder.
4. Click **Open**. SMS will open the previously created SRH-2D model and display it in the display window.

1.1 Overview of the SRH-2D Model Control Options

The project should open and appear as Figure 1. As displayed in the Project Explorer, the project contains mesh data, scatter data, map data, GIS data, and simulation data.

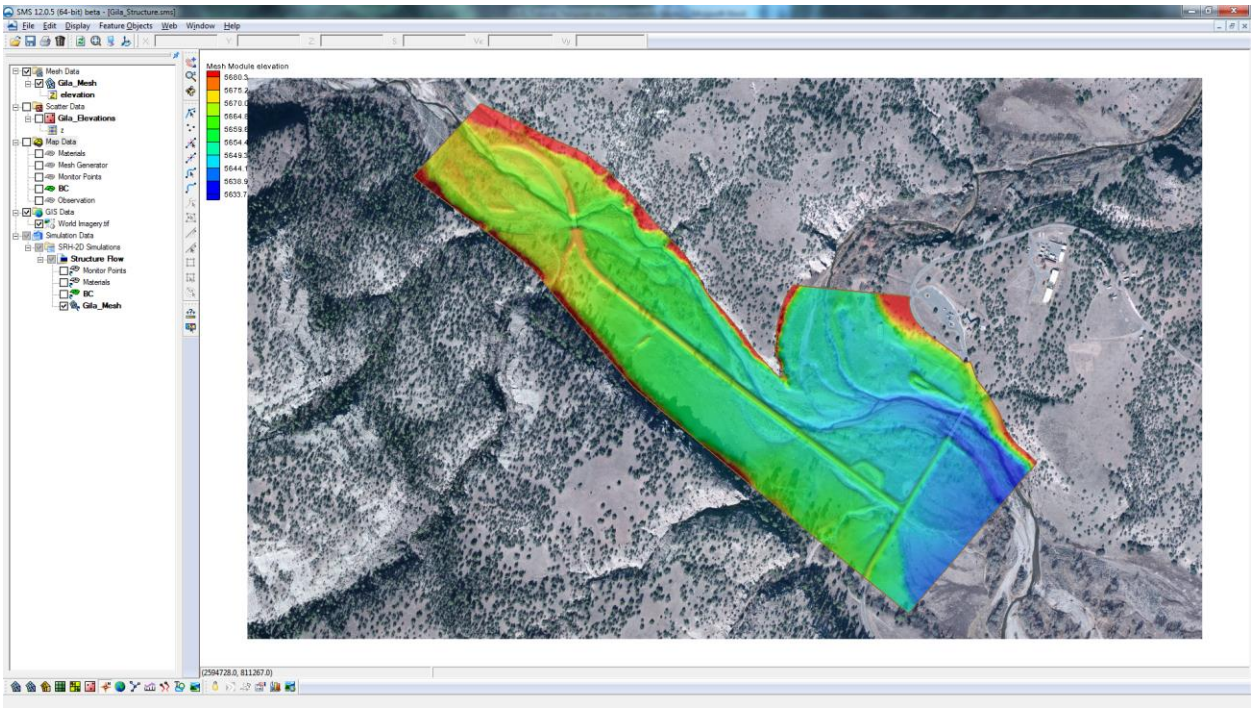


Figure 1 Existing model with simulation

1. Right-click on the "Regular Flow" simulation under "Simulation Data" in the Project Explorer and select **Model Control …** to open the *SRH-2D Model Control* dialog.

Next to review the *SRH-2D Model Control* options as they are found in the *General*, *Flow*, and *Output* tabs.

2. In the *General* tab, the options should be set to:
    - *Simulation Description*: "Gila River Inflows"
    - *Case Name*: "Standard_Run"
    - *Use temperature modeling*: unchecked
    - *Start Time (hours)*: "0"
    - *Time Step (seconds)*: "2"
    - *Total Simulation Time (hours)*: "2"
    - *Initial Conditions*: "Dry"
3. In the *Flow* tab, the options should be set to:
    - *Turbulence Model*: "Parabolic"
    - *Parabolic Turbulence*: "0.7"
4. In the *Output* tab, the options should be set to:
    - *Result Output Format*: "XMDF"

- *Result Output Unit*: "English"
- *Result Output Method*: "Specified Frequency"
- *Result Output Frequency (hours)*: "0.083333"

5. Select **OK**, to close the *SRH-2D Model Control* dialog without making any changes.
6. In the Project Explorer, review the simulation components listed under the " Regular Flow" simulation.

1.2 Overview of the SRH-2D Simulation Components

Each component listed under the simulation is a link to data found above the " Simulation Data" in the Project Explorer. Links are represented by icons with a small arrow next to them, such as the " Gila_Mesh" mesh data link and the " Monitor Points", " Materials", and " BC" map data links . As changes are made to the map and mesh data, they are immediately applied to the simulation through the link.

1. In the Project Explorer under " Map Data", turn on the display of the boundary conditions coverage by checking the box next to the " BC" coverage.
2. Select the " BC" coverage to make it the active coverage.
3. Use the **Select Feature Arc** tool and double-click on the Inlet-Q inflow boundary shown in Figure 2. This will open the *SRH-2D Linear BC* dialog.

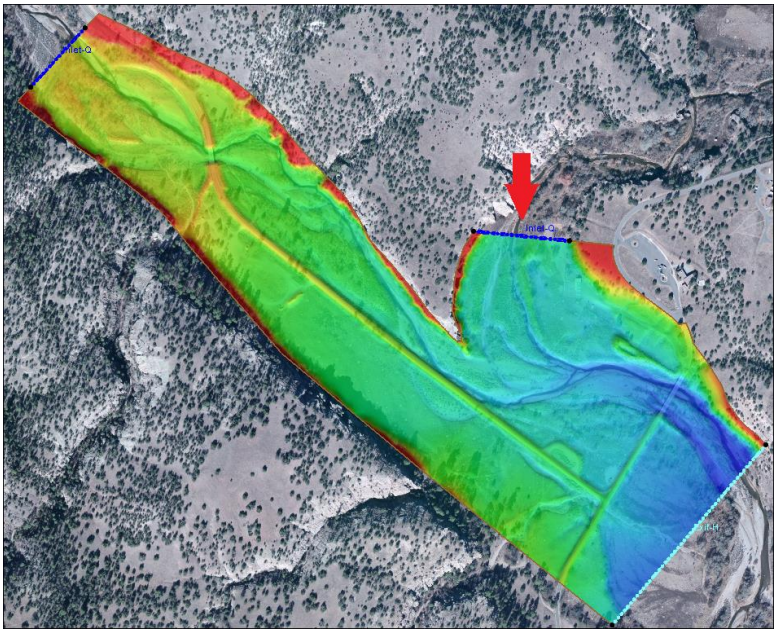


Figure 2 Inflow boundary condition

4. Click on the button under *Time Series File* to review the hydrograph that will represent the inflow. This will open the *XY Series Editor* as shown in Figure 3.

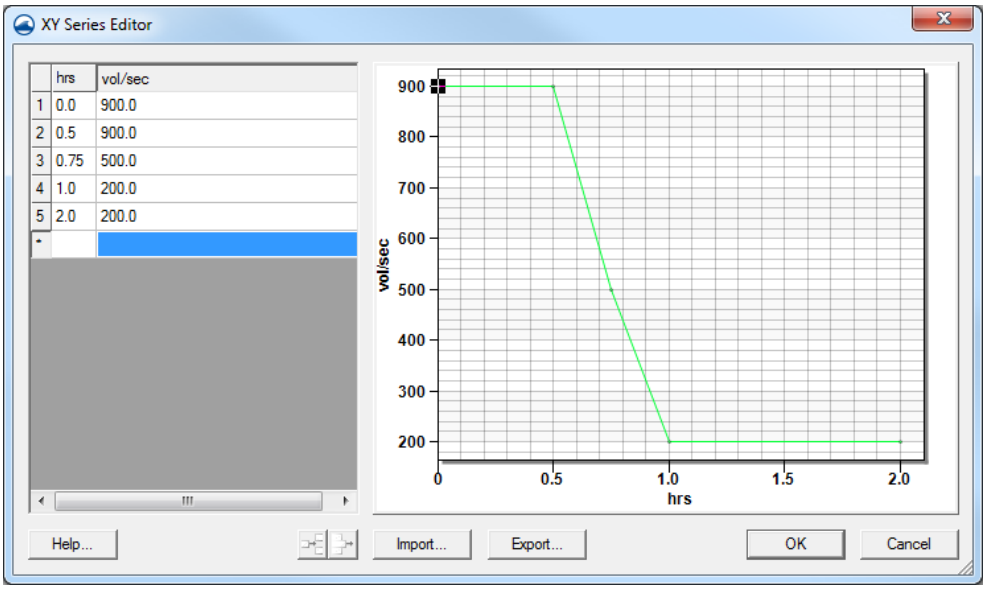


Figure 3 XY Series Editor

5. Click **OK** to close the *XY Series Editor*.
6. Click **OK** to close the *SRH-2D Linear BC* dialog.
7. Review the other Inlet-Q boundary condition as well as the Exit-H boundary condition by double-clicking on them and reviewing the inputs in the *SRH-2D Linear BC* dialog.

Within an SRH-2D model, feature points can be placed anywhere on the mesh to extract computed model data in specified areas of interest. These points are referred to as monitor points and are created within a monitor points coverage.

8. To view the monitor points, check the box next to the "Monitor Points" coverage in the Project Explorer and select it to make it the active coverage.
9. Uncheck the box next to "World Imagery.tif" to turn off the display of the background image.

A monitor point has been placed near the upstream inlet on the far left of the model and another has been placed near the outlet on the far right of the model as shown in Figure 4. When SRH-2D is run, two files in the output data will be created, one for each point. The files will contain computed model data extracted at those specific locations. Monitor points can be useful for monitoring water levels, velocities, etc. near a structure or other feature within the model.

10. Uncheck the box next to the "Monitor Points" coverage in the Project Explorer to turn off the display of the monitor points.

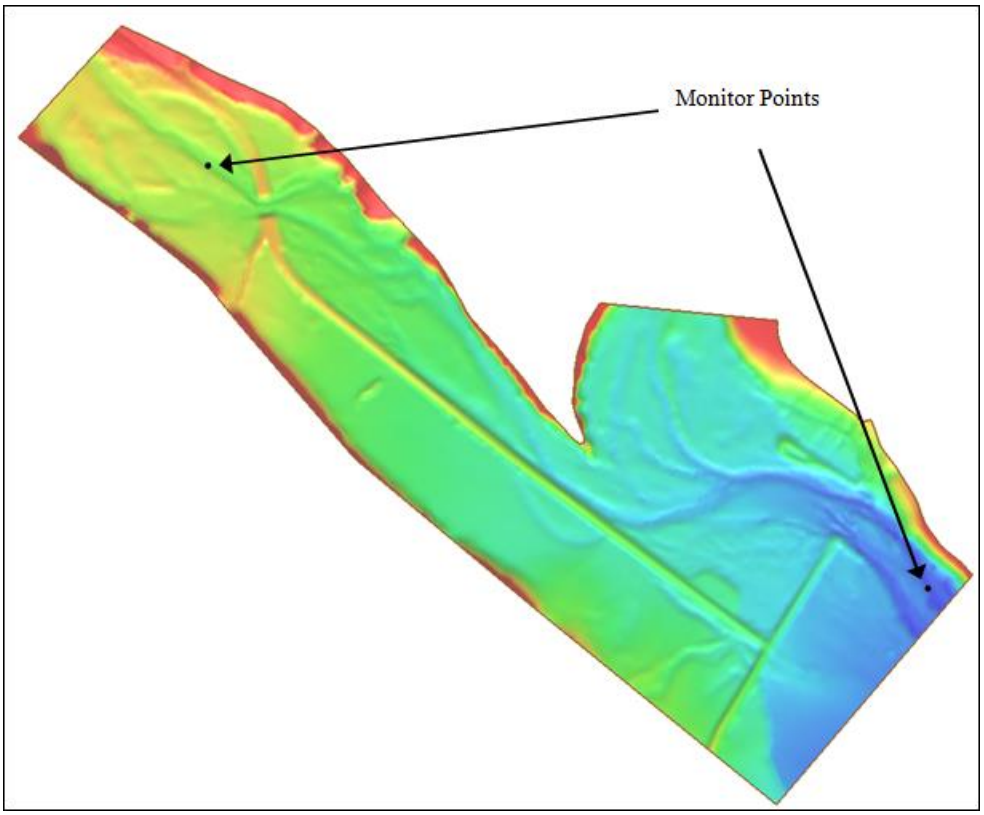


Figure 4 Monitor point locations

11. Under "Map Data" in the Project Explorer, check the box next to "Materials" and make it the active coverage.
12. Under "Mesh Data", uncheck the box next to "Gila Mesh" to turn off the display of the mesh.
13. Right-click on "Map Data" in the Project Explorer and choose **Display Options…** to bring up the *Display Options* dialog.
14. Check the box next to *Legend* and select **OK** to close the *Display Options* dialog.
15. Using the legend as a guide, observe the different material zones that have been created.
16. When finished, turn off the display of the materials by un-checking the box next to the "Materials" coverage and turn on the display of the mesh and background image by checking the boxes next to "Gila_Mesh" and "World Imagery.tif".

A finite element mesh has been created across the domain of the model and is linked to the simulation. This finite element mesh contains the geometry of the surface upon which the water will flow. It is a digital elevation model of the terrain within and around the Gila river.

17. In the Project Explorer right-click on "Mesh Data" and choose **Display Options…** to bring up the *Display Option* dialog again.

18. Check the box next to *Elements* and select **OK** to close the *Display Option* dialog and turn on the display of the elements that make up the mesh.
19. Use the **Zoom** and **Pan** tools to view the elements.
20. Right-click on "Mesh Data" and choose **Display Options…** to bring up the *Display Option* dialog again.
21. Uncheck the box next to *Elements*.
22. On the left of the *Display Options* window, select *Map*.
23. Uncheck the box next to *Legend* to turn off the display of the map legend. Select **OK** to close the *Display Options* window.

2 Running the Simulation

Next, run the simulation and load in the solution for viewing.

1. Right-click on the "Regular Flow" simulation and choose **Save, Export, and Launch SRH-2D**.

The SRH-2D model wrapper will appear and begin to run the Pre-SRH-2D application. Once that is finished, the SRH-2D window will appear and begin running.

While running SRH-2D, three DOS windows will appear. One is the SRH-2D graphic window and the other two are showing the residuals and water surface elevations at the monitor points.

2. Arrange and resize the three DOS windows so that the residual and water surface elevation plots can be seen. The arrangement may appear similar to Figure 5.

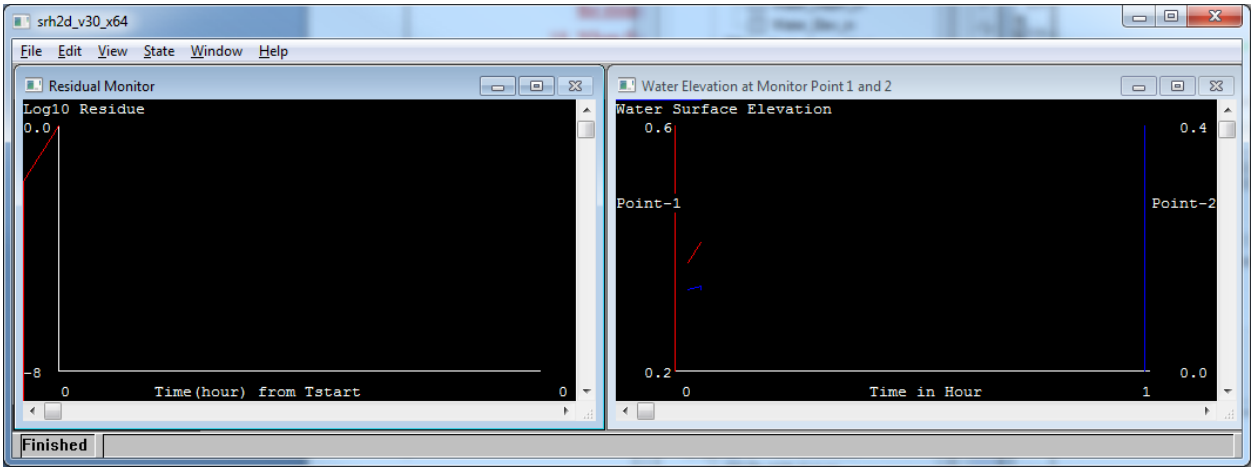


Figure 5 SRH-2D model wrapper

3. Once it has finished running, a message should appear stating: "Program terminated with exit code 0". This is the correct termination message. If another message appears, then troubleshoot the errors that appear. Click **Yes** to continue.
4. Make sure the *Load Solution* box in the SMS model wrapper is checked, as shown in Figure 6, then click **Exit**.

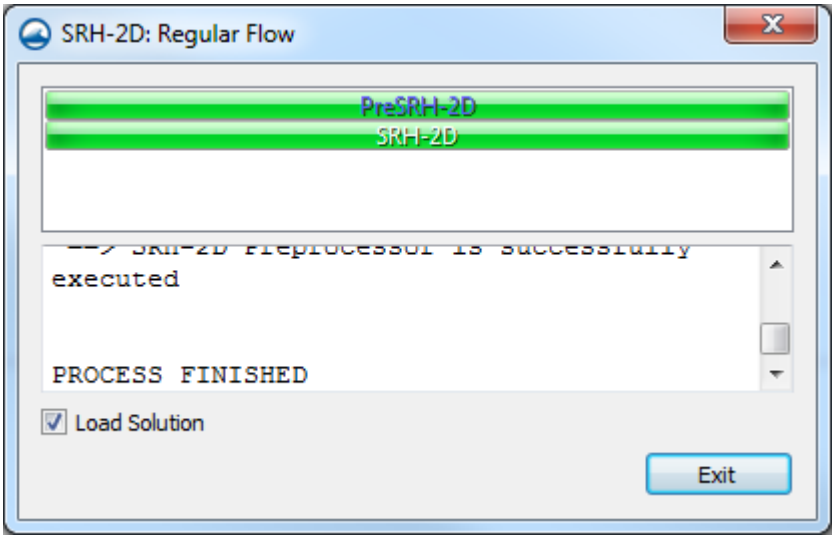


Figure 6 SMS model wrapper

5. Toggle through the datasets and time steps to observe the computed results.
6. Notice while looking at the "Water_Elev_ft" dataset that in the beginning time steps the water is backed up and begins to flow over the road as it attempts to flow through the narrow at the bridge upstream of the confluence.

Now organize the results from the simulation run by creating a folder in the Project Explorer.

7. Right-click on "Gila Mesh" in the Project Explorer and select **New Folder**.
8. Rename the folder to "Standard Run".
9. Multi-select the following datasets by holding down the *Shift* key and clicking on them: "Froude", "Strs_lb_p_ft2", "Vel_Mag_ft_p_s", "Water_Depth_ft", "Water_Elev_ft", and "Velocity".
10. Drag the selected datasets under the "Standard Run" folder that was just created.
11. Select the "Water_Elev_ft" dataset to make it active.

3 Duplicating the Simulation and Coverages

In some cases, it's necessary to make modifications to the model but the original model setup needs to be saved. Since SMS is capable of managing multiple simulations in the

same project, an easy way to accomplish this is to make a duplicate of the original simulation and make modifications to the duplicated simulation.

3.1 Duplicating the Simulation

Begin by duplicating the simulation. This will create an identical copy of the current simulation that can be used to make modifications.

1. Right-click on the "Regular Flow" simulation and select **Duplicate.**
2. Right-click on the "Regular Flow (2)" simulation and select **Rename.**
3. Rename the simulation as "Modified Flow".

3.2 Duplicating the Coverages

Slight modifications will be made to one of the inflow boundary conditions and the material properties. To make these modifications, the "BC" and "Materials" coverages will be duplicated and renamed.

1. In the Project Explorer and under *Map Data*, right-click on the "Materials" coverage and select **Duplicate**.
2. Right-click on "Materials (2)" and select **Rename** then enter "Modified Materials" as the new name.
3. Right-click on the "BC" coverage and select **Duplicate**.
4. Right-click on "BC (2)" and select **Rename** then enter "Modified BC" as the new name.

4 Unlinking/Linking Simulation Components

To update the coverages in the newly created "Modified Flow" simulation, the links to the coverages from the original simulation will need to be removed and new links will need to be created that will link the duplicated coverages to the new simulation.

1. Under the "Modified Flow" simulation, right-click on the "Materials" link and choose **Unlink**.
2. In a similar manner, right-click on "BC" under the "Modified Flow" simulation and select **Unlink**.
3. Under "Map Data" right-click on the "Modified Materials" coverage and select *Link To* | **SRH-2D Simulations → Modified Flow**.
4. In a similar manner, right-click on the "Modified BC" coverage and select *Link To* | **SRH-2D Simulations → Modified Flow**.

The new simulation is now ready for modifications.

5 Modifying the Duplicated Coverages

Now that the original simulation and coverages have been duplicated and re-linked, the flow for one of the inflow boundary conditions and one of the Manning's *n* material values will be modified within the newly duplicated simulation. SRH-2D will then be re-run and the results will be compared to the original simulation results.

1. Under "Map Data" in the Project Explorer right-click on the "Modified Materials" coverage and choose **Material Properties**.
2. In the *Material Properties* dialog, select the "Light Veg" material type from the list on the left and change the *Constant N* value to "0.075". Click **OK**.
3. In the Project Explorer, select the "Modified BC" coverage to make it active.
4. Use the **Select Feature Arc** tool to double-click on the *Inlet Q* boundary condition shown in Figure 7. This should open the *SRH-2D Linear BC* dialog.

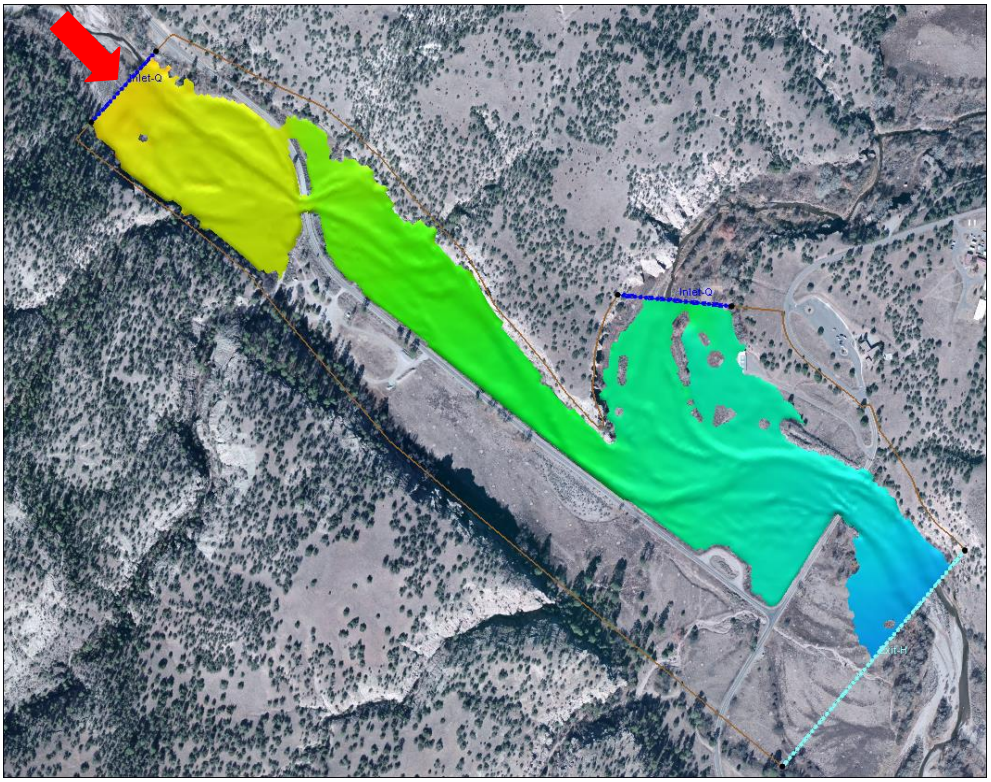


Figure 7 Inlet Q boundary condition

5. In the *Linear BC* dialog, select the curve button below *Time Series File*. This will open the *XY Series Editor* and display the hydrograph assigned to the inflow BC.
6. In the *XY Series Editor*, change the flow rate for time "0.0" and "0.5" to "5000" cfs as displayed in Figure 8.

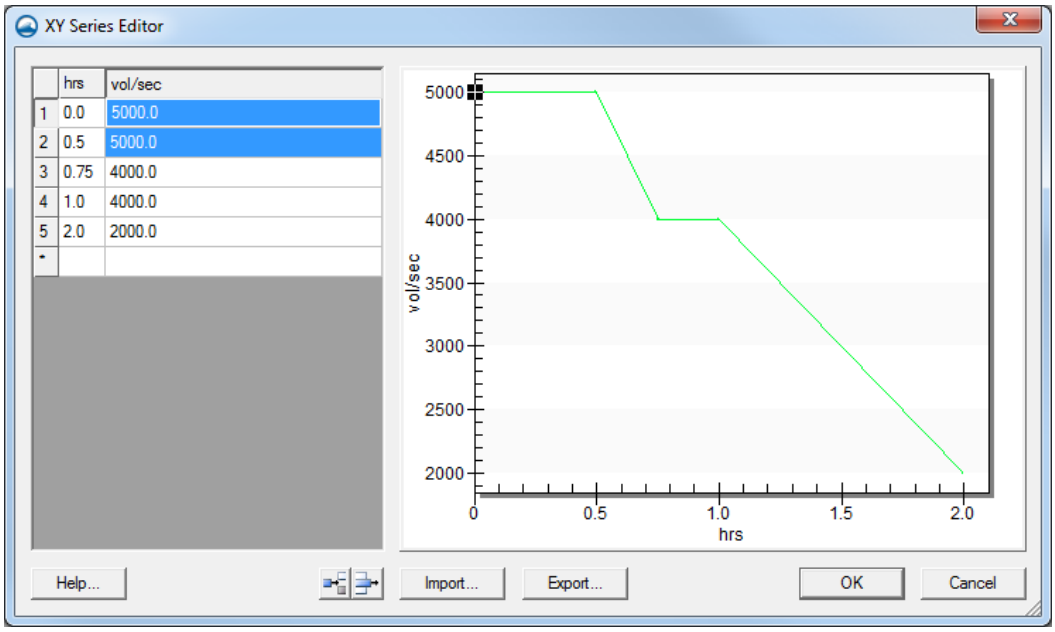


Figure 8 XY Series Editor

7. Select **OK** twice to exit the *XY Series Editor* and the *SRH-2D Linear BC* dialog.

Since the "Modified Materials" and "Modified BC" coverages are linked to the "Modified Flow" simulation, the changes that were made are current in the simulation.

6 Running the Duplicated Simulation and Comparing Results

6.1 Running the Duplicated Simulation

Now that the desired modifications have been made to the duplicated coverages, the simulation is ready to be run.

1. Right-click on the "Modified Flow" simulation and choose **Model Control…** to bring up the *SRH-2D Model Control* dialog.
2. In the dialog, change the *Case Name* to be "Modified_Run".
3. Select **OK** to exit the *SRH-2D Model Control* dialog.
4. Right-click on the "Modified Flow" simulation again and choose **Save, Export, and Launch SRH-2D**.
5. When SRH-2D has finished running, select **Yes** to exit the SRH-2D model run window.
6. Make sure *Load Solution* is checked on in the model wrapper window and choose **Exit**.

To better organize the solution results, a folder in the mesh data will be created as with the original simulation run.

7. In the Project Explorer, right-click on the "Gila_Mesh" mesh and choose **New Folder**.
8. Name the folder "Modified Run".
9. Multi-select the new solution datasets by holding down the *Shift* key and clicking on them: "Froude", "Strs_lb_p_ft2", "Vel_Mag_ft_p_s", "Water_Depth_ft", "Water_Elev_ft", and "Velocity".
10. Drag the selected datasets under the "Modified Run" folder. When finished the dataset organization should look like Figure 9.

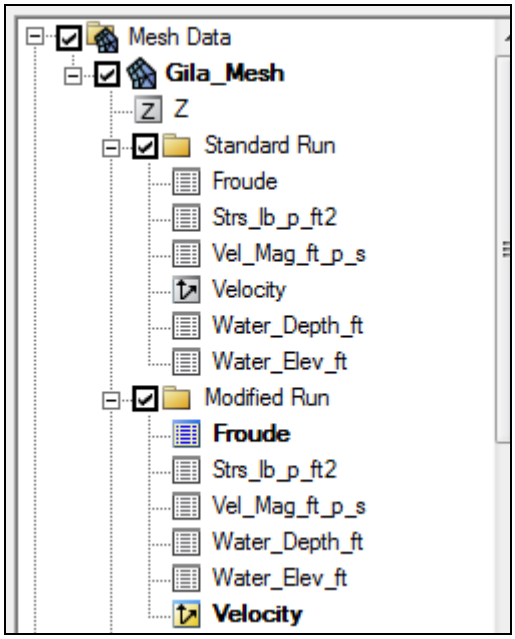


Figure 9 Mesh dataset organization

11. Toggle back and forth between the two different simulation results datasets and time steps to compare differences.

6.2 Comparing Results

SMS has several data tools for visualizing the results and comparing differences. One simple comparison could be to use the *Dataset Calculator* to create a dataset showing the difference in velocity magnitudes for the two simulation results as demonstrated in the following steps.

1. Select *Data* | **Dataset Toolbox…** to bring up the *Dataset Toolbox.*

2. In the dialog, select the "Data Calculator" from the list of *Tools*, then select "Vel_Mag_ft_p_s" dataset under the "Standard Run" folder and check the box to *Use all time steps*.
3. Select the **Add to Expression** button.
4. Select the subtract button.
5. Select the "Vel_Mag_ft_p_s" dataset under the "Modified Run" folder and select **Add to Expression**.
6. Rename the *Output dataset name* to "Velocity_Diff" and select **Compute**. A new dataset should appear under "Mesh Data".
7. Select **Done** to exit the *Dataset Toolbox* window.
8. Ensure that the newly created "Velocity_Diff" dataset is active and toggle through the time steps to see the differences in the contours.

The majority of the mesh will show contours representing zero difference in velocity magnitudes, however changes can be seen in and near the channel. Contour options can be adjusted as desired to better view the differences.

Further comparisons can be made on the data as desired.

7 Conclusion

This concludes the "SRH-2D-Working with Simulations"[1] tutorial. Topics covered in this tutorial included:

- An overview of the SRH-2D simulation structure.
- Reviewing and running an SRH-2D simulation.
- Duplicating simulations and coverages.
- Linking/Unlinking simulation components.
- Saving a project file.

Continue to experiment with the SMS interface or quit the program.

[1] This tutorial was developed by Aquaveo, LLC under contract with the Federal Highway Administration.