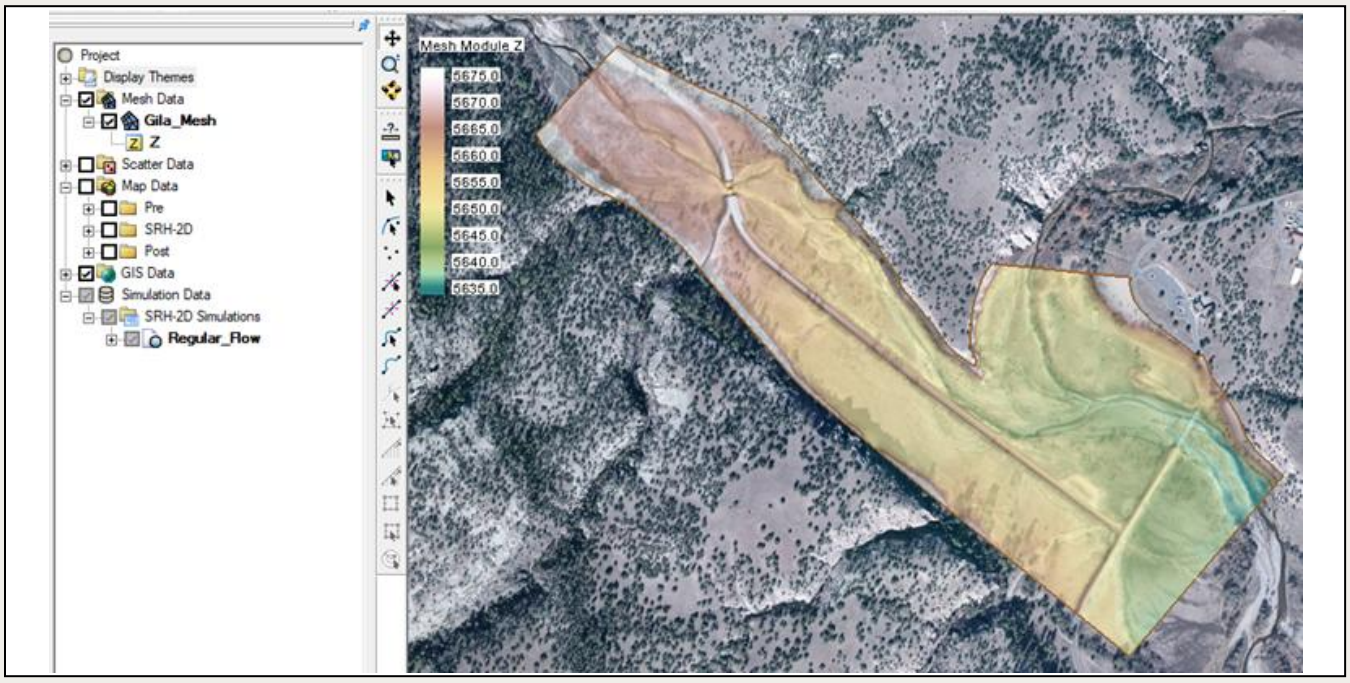
SRH-2D Tutorial

Simulations

Creating multiple SRH-2D simulations in SMS



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.

Prerequisite Tutorials

- SRH-2D

Required Components

- SMS Core
- SRH-2D Model & Interface

Time

- 15–20 minutes

1 Introduction

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 simulation 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. Refer to the appropriate SMS which demonstrate the creation of hydraulic structures within this SRH-2D model.

1. Open a new instance of SMS.
2. Select *File* | **Open**… to bring up the *Open* dialog.
3. Navigate to the *data files* folder for this tutorial and select the file “Gila_Base.sms”.
4. Click **Open** to import the file and open it in the display window.

The project should open and appear as Figure 1.

Figure 1 Existing model of Gila River

1.1 Overview of the SRH-2D Simulation

As displayed in the Project Explorer, the project contains display themes, a mesh, a background scatter set, various map coverages representing the mesh generation process and the SRH-2D simulation, a background aerial photo, and an SRH-2D simulation.

A mesh has been created across the domain of the model and is linked to the simulation. This 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. To review the conditions represented by this simulation, examine both the mesh and the model parameters.

1. In the Project Explorer, right-click on "Mesh Data" and choose **Display Options…** to bring up the *Display Option* dialog.
2. Check the box next to the *Elements* option.
3. Select **OK** to close the *Display Option* dialog and display the mesh elements.
4. Use the **Zoom** and **Pan** tools to view the elements.

Note the smaller elements (higher resolution) along the river bed, the quadrilateral elements along the roadways, and the hole in the mesh where the bridge pier blocks flow.

5. Right-click on "Mesh Data" and choose **Display Options…** to bring up the *Display Option* dialog again.
6. Uncheck the box next to the *Elements* option.
7. Select **OK** to close the *Display Option* dialog.

The mesh generation process can be reviewed in other tutorials.

To review the simulation, do the following:

8. Under "Simulation Data", right-click on "Regular Flow" and select **Model Control…** to open the *SRH2D Model Control* dialog.
9. In the *General* tab, the options should be set to:
    - *Simulation description*: "Gila River Inflows"
    - *Case name*: "Standard_Run"
    - *Start time (hours)*: "0.0"
    - *Time step (seconds)*: "2.0"
    - *End time (hours)*: "2.5"
    - *Initial condition*: "Automatic"
10. In the *Output* tab, the options should be set to:
    - *Output format*: "XMDF"
    - *Output units*: "English"
    - *Output method*: "Simulation End"
11. In the *Advanced* tab, the options should be set to:
    - *Turbulence model*: "Parabolic"
    - *Parabolic turbulence*: "0.7"

- *Unsteady output* should be unchecked (off)

12. Select **OK** to close the *SRH2D Model Control* dialog.

1.2 Overview of the SRH-2D Simulation Components

Each component listed under the "Regular Flow" 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", "Materials", and "BC" map data links . The components have been applied to the simulation.

1. Under "Map Data", expand the "SRH-2D" folder and check the box next to the "BC" coverage to display the boundary conditions coverage.
2. Select the "BC" coverage to make it active.

The assigned boundary conditions will be identified similar to Figure 2. Note there are two inflow boundaries and one downstream water surface elevation boundary. Refer to the "SRH-2D" tutorial to review both how these boundaries can be specified and the supported options.

3. Use the **Select Feature Arc** tool and double-click on any of the boundary arcs.

This will open the *SRH2D Assign BC* dialog and display the specified boundary type and value.

4. Click **OK** to close the *SRH2D Assign BC* dialog.
5. If desired, review the other boundary conditions in the same manner.

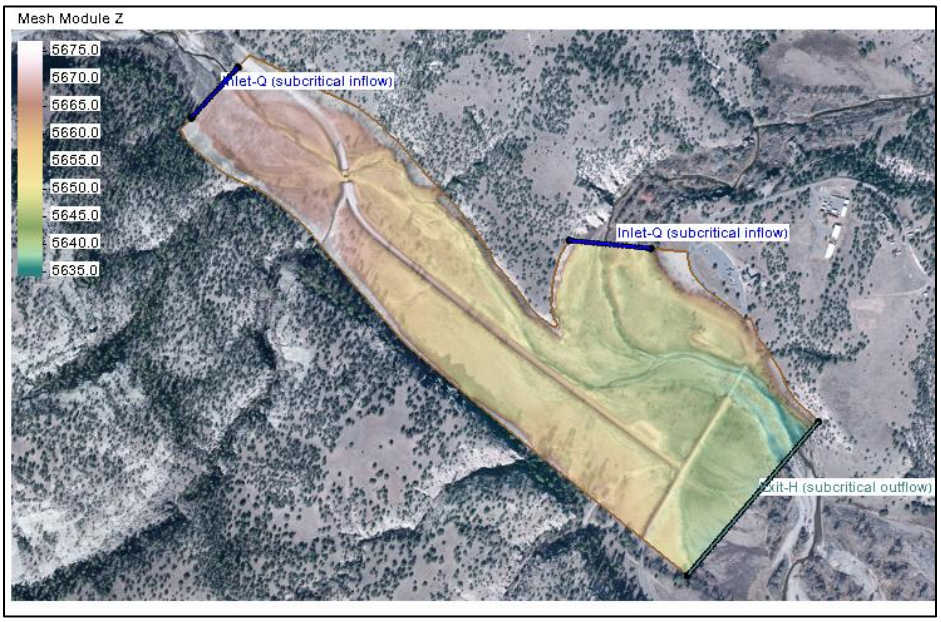


Figure 2 Boundary condition

Within an SRH-2D model, monitor points and lines can be placed anywhere in the mesh to extract computed model data in specified areas of interest.

6. To view the monitor lines and points, check the box next to the "Monitor" coverage and select it to make it active.

Monitor points have been placed near the upstream end (far left of the model) of the simulations and near the downstream end (on the far right of the model) as shown in Figure 3. SRH-2D creates files for each point as it runs that contain computed water and ground levels at those locations. Monitor arcs (lines) have been placed near the boundary conditions and upstream and downstream of the bridge. SRH-2D creates files for each monitor line containing flux and water levels computed along those lines.

7. Uncheck the "Monitor" coverage to turn off the display.

Figure 3 Monitor object locations

8. Check the box next to "Materials" and make it the active coverage.
9. Under "Mesh Data", uncheck the box next to "Gila Mesh" to turn off the display of the mesh.
10. Uncheck the box next to "GIS Data" to turn off the aerial image to make the material display more visible.
11. Right-click on "Map Data" and choose **Display Options…** to bring up the *Display Options* dialog.
12. Check the box next to *Legend.*
13. Click **OK** to close the *Display Options* dialog.
14. Using the legend as a guide, observe the different material zones that have been created and the associated Manning's N values.
15. In the Project Explorer, right-click on "Map Data" and choose **Display Options…** to bring up the *Display Options* dialog again.
16. Uncheck the box next to *Legend* to turn off the display of the map legend.
17. Select **OK** to close the *Display Options* window.
18. When finished, turn off the display of the materials by un-checking the box next to the "Materials" coverage.

19. Check the boxes next to "Gila_Mesh" and "GIS Data" to turn on the display of the mesh and background image.

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 Project, Simulation and Run**.
2. If a warning comes up saying the previous solution will be unloaded, click **OK**.

SMS will save the project and then the simulation. Progress will be displayed in the *SRH-2D Export Simulation* dialog, which will automatically close if no errors are encountered. (Note: this only takes a couple of seconds.)

The *Simulation Run Queue* dialog will then appear and begin to run the Pre-SRH-2D application. Once that is finished, the SRH-2D application will begin.

While running SRH-2D, progress is shown in the *Simulation Run Queue* dialog.

3. In the top portion of the *Simulation Run Queue* dialog, under the "Regular Flow" item, select the "SRH-2D" item as showing in Figure 4.

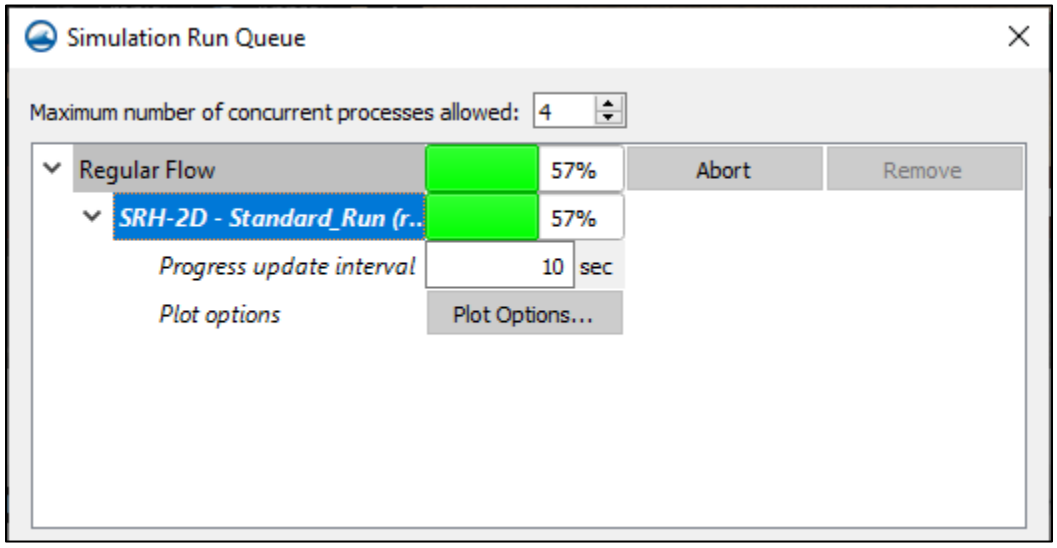


Figure 4 SRH-2D Simulation Run Queue

4. Highlight the "SRH-2D" item to populate the lower portion of the dialog.
5. Select the *Mass Balance Plot* tab.

This will populate with a plot of the mass balance as the run proceeds. Each tab of the run queue displays a different plot (or information) about the progress and stability of the simulation. Review the "Simulation Plots" tutorial for more information.

6. When the simulation run has finished, click the **Load Solution** button to import the results of the simulation run and to clear and close the *Simulation Run Queue* dialog.
7. Click on the "Vel_Mag_ft_p_s", "Water_Depth_ft", and "Water_Elev_ft" datasets in turn.

Note that each has a display theme associated with the dataset name. Refer to the "Display Themes" tutorial for more information on this feature. Also note that the solution

shows the extents of the flooded areas. Specifically, make note that the water is backed up behind the bridge and flows over the road to the left of the river. Also, the inflow of the tributary is split into two channels.

3 Duplicating the Simulation and Coverages

Generally, in the process of simulating an area, it is necessary to make multiple variations of the simulation. Best practices dictate that the original model setup be saved. SMS is capable of managing multiple simulations in the same project, so an easy way to accomplish this is to make a duplicate of the original simulation and make modifications to the duplicated simulation.

As noted above, the flow entering the domain at the tributary includes a split flow condition. Since this user specified location and orientation of the inflow boundaries can have significant influence on results, this section of the tutorial will demonstrate how to compare this solution with an alternative boundary specification.

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. Type "Modified_Flow" and press *Enter*.

Note: the solution for the simulation is not included in the duplicate.

3.2 Duplicating the BC Coverage

To illustrate this feature, modifications will be made to the location of the tributary inflow boundary. To make these modifications, the "BC" coverages will be duplicated and renamed.

1. Under "SRH-2D", right-click on "BC" and select **Duplicate**.
2. Right-click on "BC (2)" and select **Rename**.
3. Enter "Modified BC" as the new name.

4 Switching 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 "Modified_Flow", right-click on "BC" and select **Remove** to remove the original boundary conditions from the new simulation.
2. Right-click on "Modified BC" and select *Apply to* | **SRH-2D Simulations → Modified_Flow**.

This links the new boundary conditions to the simulation. The new simulation is now ready for modifications.

5 Modifying the Duplicated Boundary Condition Coverage

Now that the original simulation and coverage have been duplicated and re-linked, the position of the inflow boundary condition will be modified in the new BC coverage.

1. In the Project Explorer, select "Modified BC" to make it active.
2. **Zoom** in on the tributary inflow boundary condition (Inlet-Q).

Figure 5 shows a reduced boundary condition arc. To shorten the existing arc to restrict the inflow to a single channel as shown in Figure 5, do the following:

3. Use the Select Feature Node tool to select and drag the end nodes of the arc to the positions shown in Figure 5.
4. Use the Select Feature Vertex tool to select and delete the vertices at the ends of the arcs, beyond the new nodes.

This will restrict the inflow for the tributary to this single channel.

Figure 5 Inlet Q boundary condition

Note: It is not required to duplicate boundary condition or material coverages and simulations to consider just changes in the values. This can be done using the scenario feature of an advanced simulation. The "SRH-2D Advanced Simulation – Scenarios" tutorial describes these options.

6 Running the Modified Simulation and Comparing Results

For bookkeeping purposes, it may be useful to update the metadata for the modified simulation. To do this, and then to run the modified simulation:

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 *SRH2D Model Control* dialog.
4. Right-click on the "Modified_Flow" simulation and choose **Save Project, Simulation and Run**.
5. If a warning comes up saying the coverages will be renumbered, click **Yes**.
6. When the simulation has finished running, click **Load Solution** to import the results and to clear and close the *Simulation Run Queue* dialog.
7. Click **Close** to exit the *Simulation Run Queue* dialog.
8. Click back and forth between the two simulation results datasets to compare differences.

In this case, it is obvious that the change to the boundary condition location did not have much of an impact. If a quantitative difference is required, SMS has several data tools for visualizing the results and comparing differences. One simple comparison could be to use the *Dataset Calculator* in the **Toolbox** to create a dataset showing the difference depth or velocity magnitude. The "Data Visualization" and "SRH-2D Post-Processing" tutorials illustrate how to compare simulation results.

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.