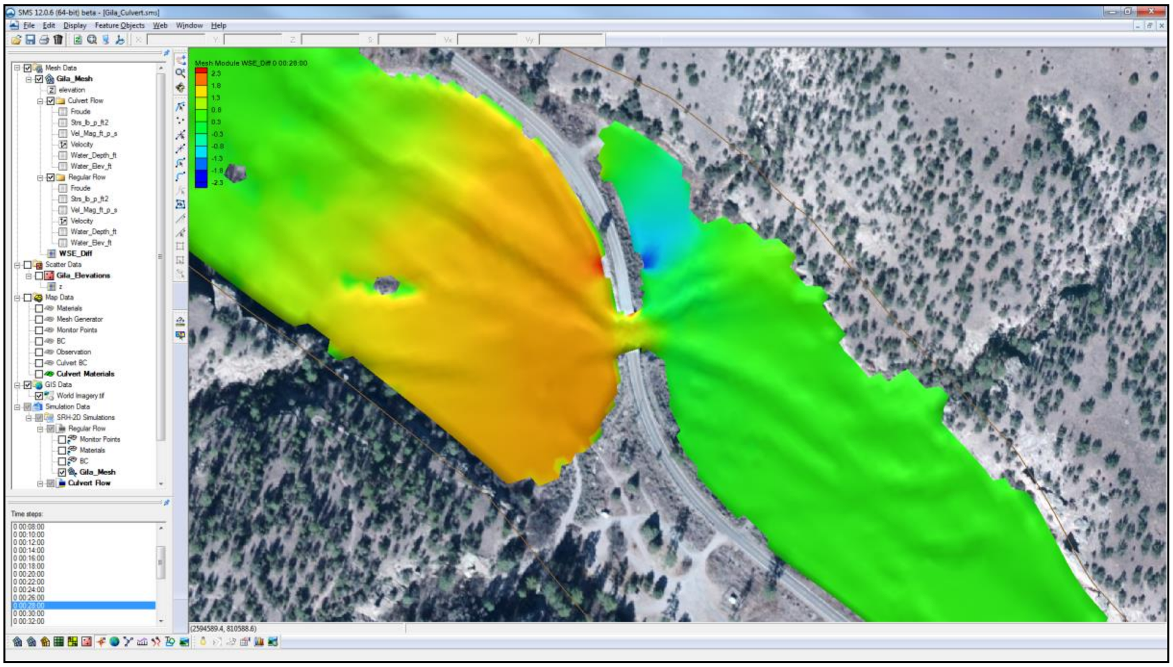
U.S. Department of Transportation
Federal Highway Administration

SMS v. 12.0

SRH-2D Tutorial

Culvert Structures



Objectives

This tutorial demonstrates the process of modeling culverts in SRH-2D. The "Working with Simulations" tutorial should have been completed before attempting this one. All files for this tutorial are found in the "Data" folder within the " SRH2D_Culvert" folder.

Prerequisites

- SRH-2D-Working with Simulations

Requirements

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

Time

- 25-30 minutes

AQUAVEO™

1 Model Overview

An existing SRH-2D model will be used to facilitate the setup for this tutorial. SRH-2D provides two different ways to define a culvert. One way couples the FHWA HY8 culvert model with SRH-2D and the other way utilizes the culvert definition built into SRH-2D. This tutorial will demonstrate the latter.

The area being modeled is located at the confluence of the West and Middle forks of the Gila River, located in New Mexico. During high flows, a significant amount of water is backed up near one of the roadway bridges causing flooding upstream. The purpose of this tutorial is to simulate a culvert relief structure near the bridge to mitigate the flooding.

Although the culvert structure is a capable option for modeling culverts, there are limitations to its use. Momentum calculated in the 2D computations does not transfer through the structure. Due to the nature of the culvert computations, the computational timestep is most often required to range between 0.25 and 0.5 seconds. Also, reverse flows through the culvert are not possible.

2 Getting Started

To begin, do the following:

1. Open a new instance of SMS.
2. Select *File* | **Open** to bring up the *Open* dialog.
3. Navigate to and open the “Gila_Structure.sms” project found in the “Data Files” folder for this tutorial and click **Open**.

The existing project will open and appear as displayed in Figure 1.

In the Project Explorer, duplicates of the “Regular Flow” simulation, “BC” coverage, and “Materials” coverage have been made to expedite the model setup process. The duplicates have been renamed as “Culvert Flow”, “Culvert BC”, and “Culvert Materials” respectively. The culvert structure will be created within the duplicated coverages and simulated in the “Culvert Flow” simulation.

The process of duplicating these items was demonstrated in the “Simulations” tutorial. Creating duplicates of simulations or coverages allows making modifications to a model while still preserving the original simulation or coverages. This also enables creating several modeling scenarios in the same project and comparing the solutions.

If desired, review the “Simulations” tutorial before continuing.

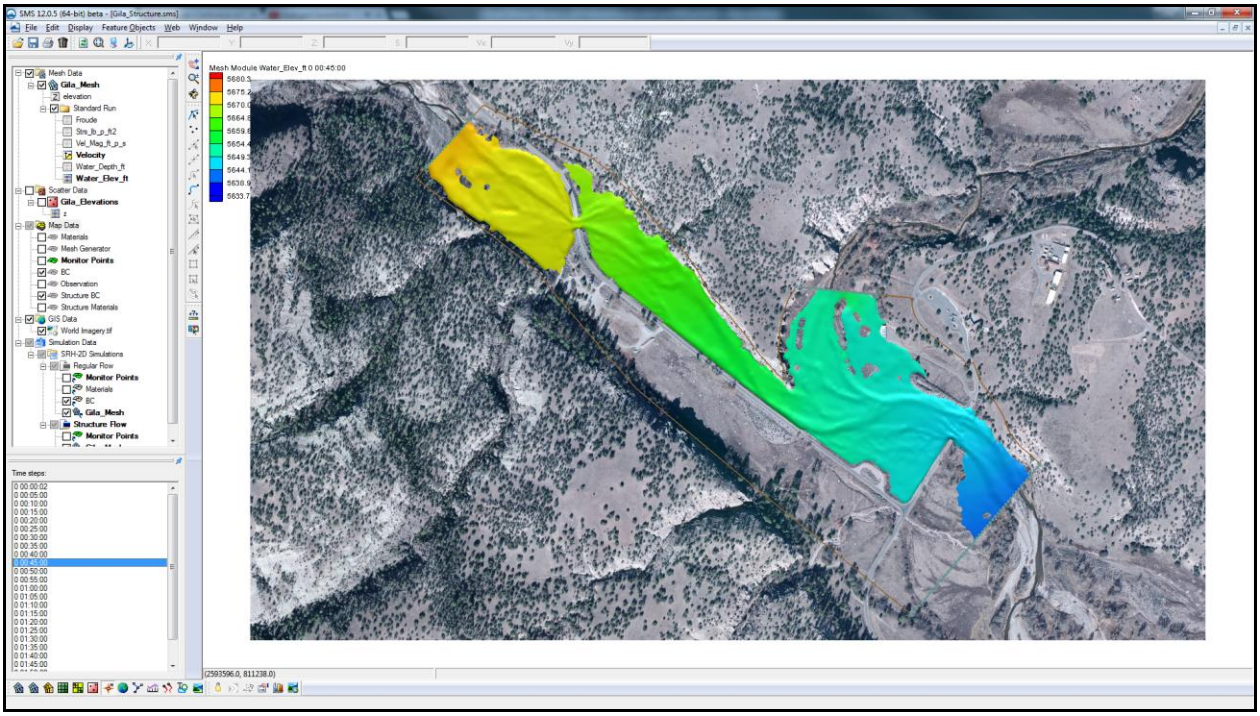


Figure 1 Gila_Structure.sms Project

The mesh datasets located under the “Standard Run” folder in the Project Explorer are from an SRH-2D solution of the existing flow conditions, without the culvert relief structure. The datasets can be used to make comparisons and visualize the effects the culvert structure boundary condition will have on the model.

3 Creating the Culvert Boundary Conditions

The culvert boundary condition will be created near the bridge location just upstream of the confluence (location displayed in Figure 2). Culvert boundary conditions are defined by creating two arcs, one on the upstream face and one on the downstream face of the structure. The arcs are then defined as a culvert structure and the attributes of the culvert are defined in the culvert definition dialog.

3.1 Creating the Boundary Condition Arcs

The first step for creating a culvert boundary condition in SMS is to create arcs representing the structure within the SRH-2D boundary condition coverage.

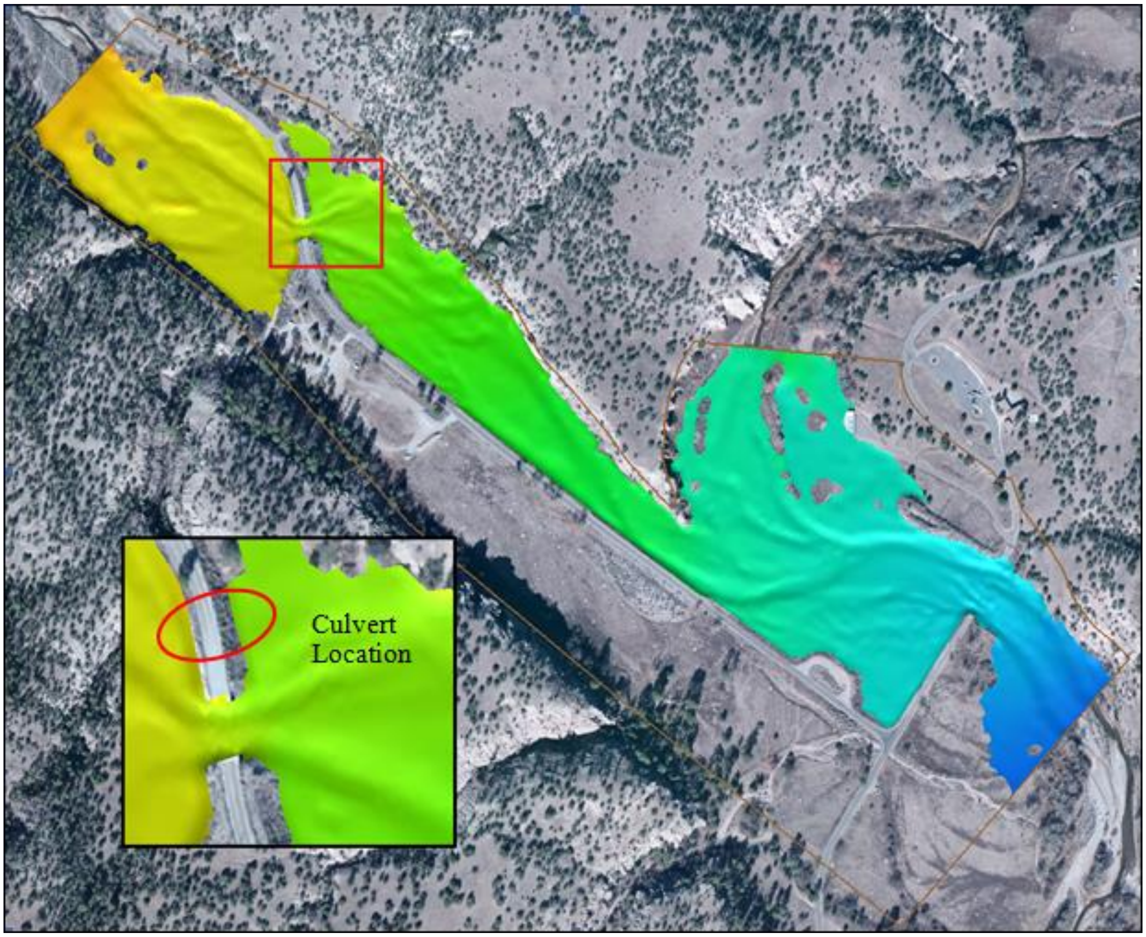


Figure 2 Culvert Location

1. Use the **Zoom** tool to zoom into the culvert location near the bridge.
2. Select the "Elevation" dataset under "Gila_Mesh" in the project explorer to display the mesh elevations.
3. Select *Display* | **Display Options…** to open the *Display Options* dialog.
4. In the *2D Mesh* section, check the box next to *Elements* to turn on the display of mesh elements. Select **OK** to exit the *Display Options*.
5. In the Project Explorer, select the "Culvert BC" coverage to make it the active coverage.
6. Use the **Create Feature Arc** tool to create one arc on each side of the road. These arcs will define the upstream and downstream faces of the culvert boundary condition. The created arcs should look similar to Figure 3.

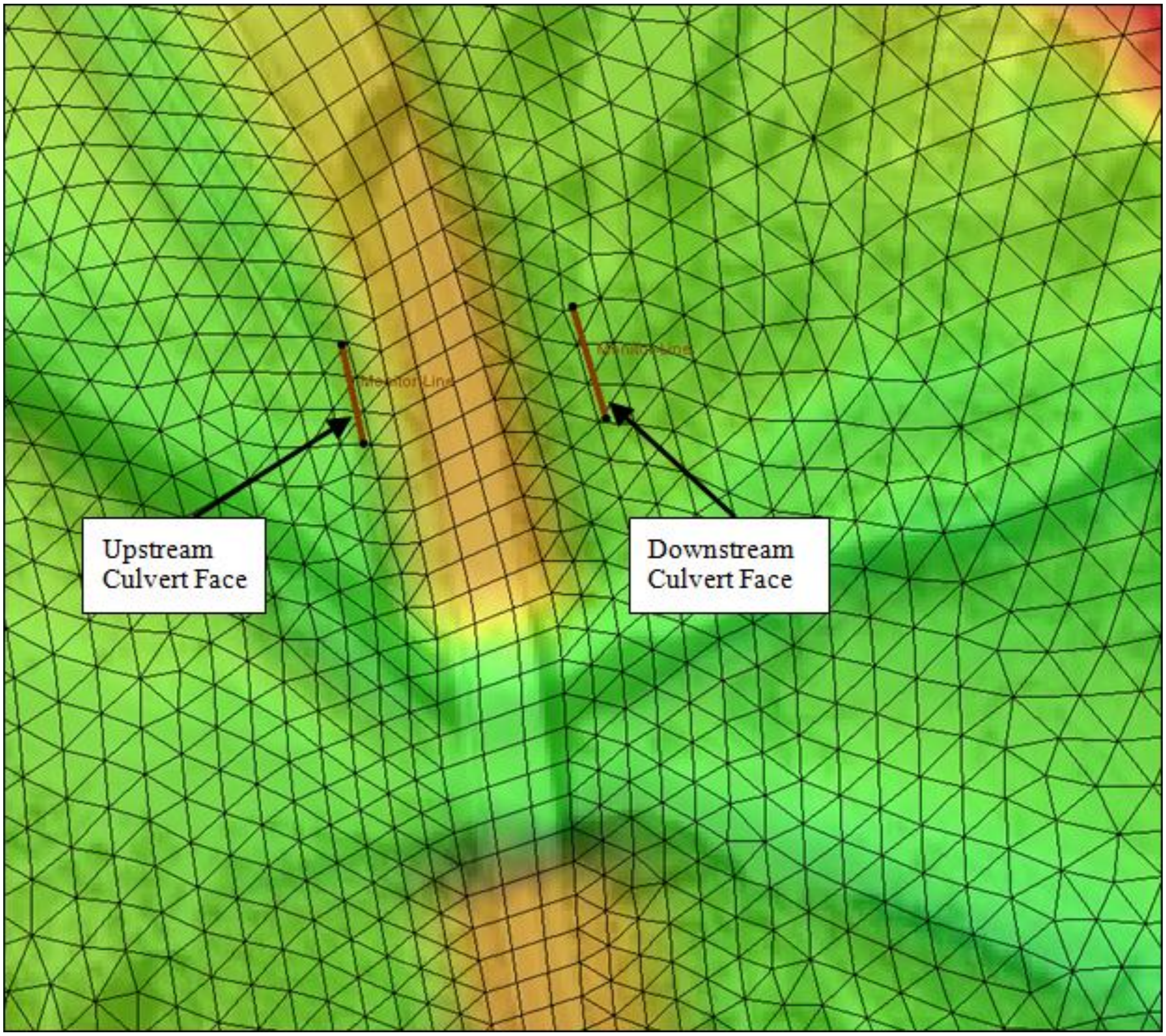


Figure 3 Upstream and Downstream BC Culvert Arcs

7. For consistency, change the coordinate locations of each of the 4 end nodes on the arcs to the coordinates shown in Figure 4. To do this, use the **Select Feature Point** tool to select a point at the end of an arc and manually enter the X and Y values into the boxes at the top of the SMS window.

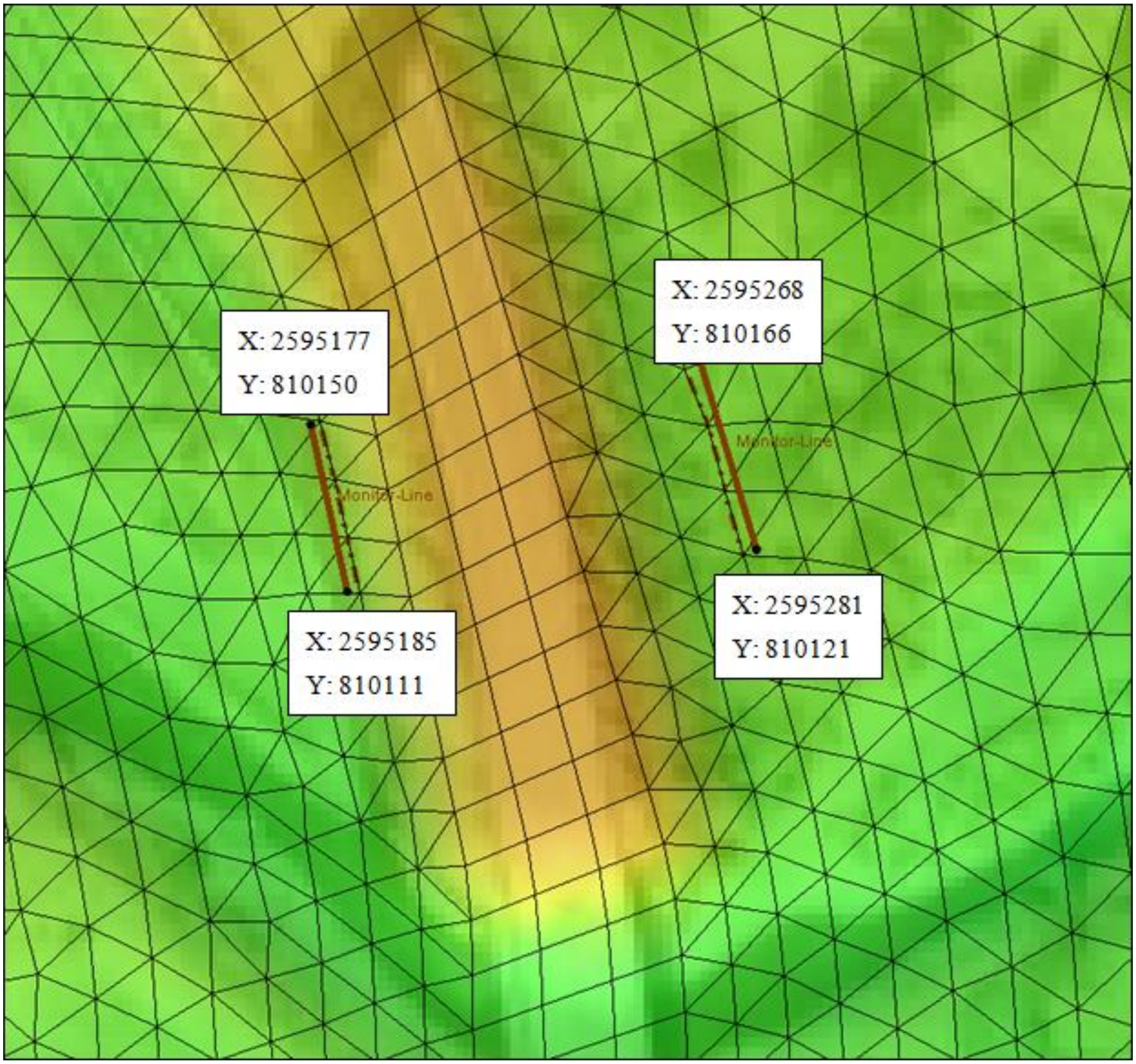


Figure 4 Coordinate Location of Arc End Nodes

Note: If any vertices exist along the arcs, they can be selected and deleted using the **Select Feature Vertex** tool and pressing the *Delete* key.

The dotted lines displayed next to the arcs represent a snapping preview of how the arcs will snap to the mesh to create a nodestring. They serve as a guide to visualize the location on the mesh where the culvert boundary condition arcs will be applied.

3.2 Assigning the BC Attributes and Defining the Culvert

The next step in creating a boundary condition is to specify the boundary condition type and define the culvert attributes.

1. Using the **Select Feature Arc** tool, select the upstream (leftmost) arc. Take note of the ID for this arc, which is displayed at the bottom of the SMS application.
2. Hold the *Shift* key and select the downstream arc so that both of the arcs are selected.
3. Right-click on either arc and select the **Assign Linear BC...** command. SMS will bring up the *Linear BC* dialog.
4. In the *Type* combo box, select "Culvert". Be sure to select "Culvert" and not "Culvert HY-8".
5. Note the assignment of "Culvert Upstream" and "Culvert Downstream" to the two arcs, associated with their ID values. If the ID displayed for culvert upstream is not the same as noted above in step 1, switch the associations using the combo box for *Role*.
6. Define the culvert and crossing attributes as found in Table 1. When done the dialog should resemble Figure 5.
7. When done, click **OK** to close the *Linear BC* dialog. This particular culvert crossing contains five concrete box barrels with straight headwalls. Definitions of the input parameters can be found at www.xmswiki.com.

Parameter	Value	Parameter	Value
Type	Culvert	Number of identical barrels	5
Upstream invert elevation (ZI)	5664.5	Entrance type (m_in)	Non-mitered
Interior height of barrel (Dc)	6	Culvert inlet coefficients (Kp, M, cp, Y)	Concrete – Rectangular – Headwall; ¾ in chamfers
Length of barrel (Dc)	85	Entrance loss coeff Ke	0.5
Area of barrel (Ac)	48	Manning roughness coefficient in barrel (Nc)	0.012
Hydraulic radius of barrel (Rh)	1.714	Crest elevation	5672
Slope of barrel (Slp)	0.0176	Length of Weir over Culvert	60
Units	ft	Type	paved

Table 1 Linear BC attributes

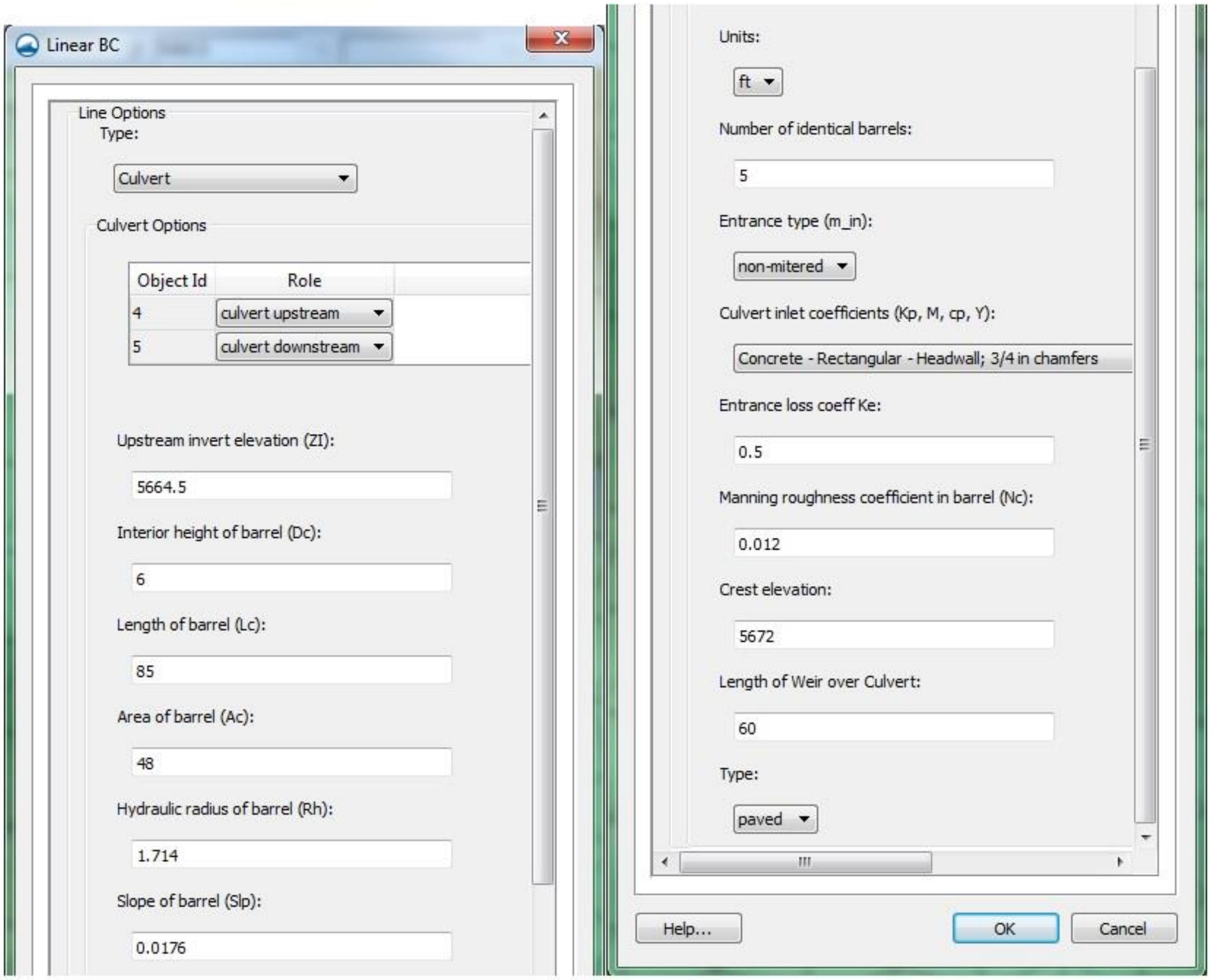


Figure 5 Culvert Parameters

3.3 Modifying the Materials Coverage

In order to properly define a culvert Boundary Condition in SMS, the materials coverage must also be modified. SMS requires that the material type of the elements between the two culvert faces be specified as “Unassigned”. Any element with an unassigned material type will be defined as a “No-flow” or inactive element.

1. In the Project Explorer turn off the display of the mesh and background image by un-checking the box next to “Mesh Data” and also the box next to “GIS Data”.

2. Turn on the display of the materials by checking the box next to the “Culvert Materials” coverage and select it to make it the active coverage.

3. Using the **Create Feature Arc** tool, draw one or more arcs enclosing the area between the two culvert arcs. If drawn correctly, the arc should close upon itself and appear similar to Figure 6.

The culvert arcs within the inactive "Culvert BC" coverage should still be visible in a dimmed forest green color and can be used as a guide when drawing the enclosed area.

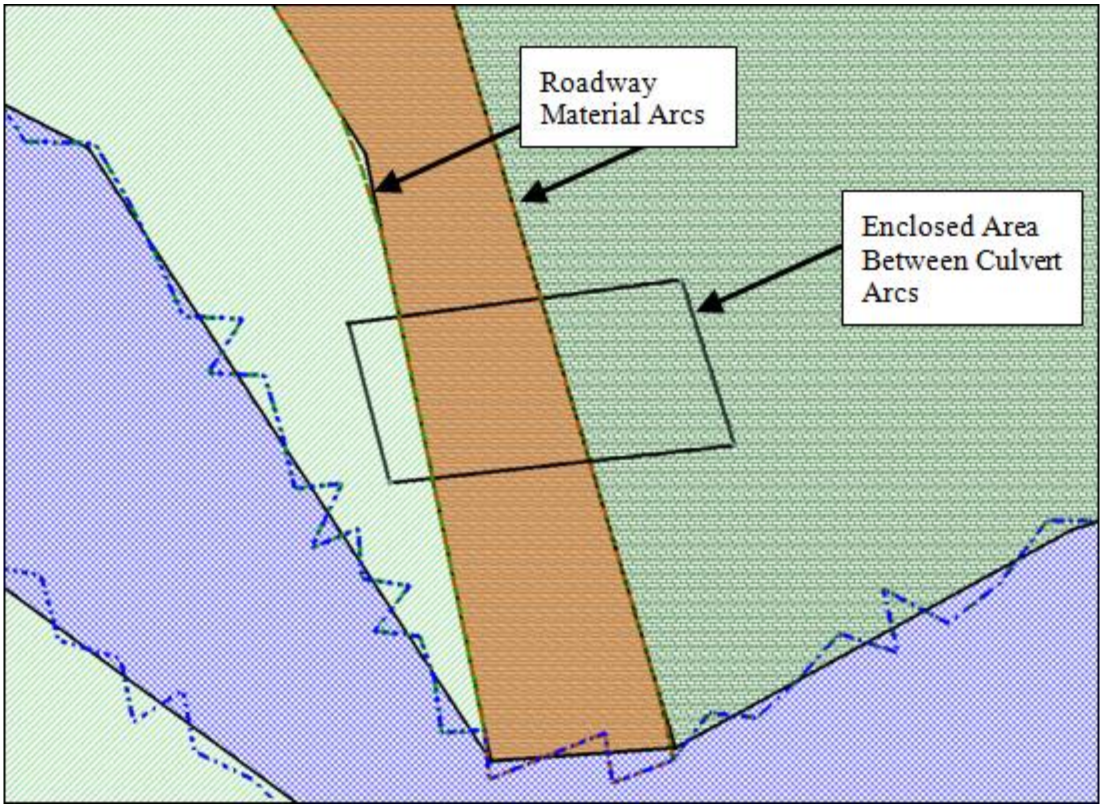


Figure 6 Enclosed Area Between Culvert Arcs and Roadway Material Arcs

4. Select *Feature Objects* | **Build Polygons**. This will create three polygons within the enclosed area between the culvert arcs.

5. Using the **Select Feature Polygon** tool multi-select the three polygons within the enclosed area by holding down the *Shift* key and selecting them.

6. Right-click on one of the selected polygons and choose **Assign Material Properties** to bring up the *Assign Material Properties* dialog.

7. Select "Unassigned" for the material type and click **OK** to close the *Assign Material Properties* window.

After making all the preceding changes to the "Culvert Materials" coverage, it should appear similar to Figure 7. The dotted lines are from the *Snap Preview* and can be turned off by pressing *Shift + Q*. The snap preview simply shows how the materials will be assigned to the mesh elements.

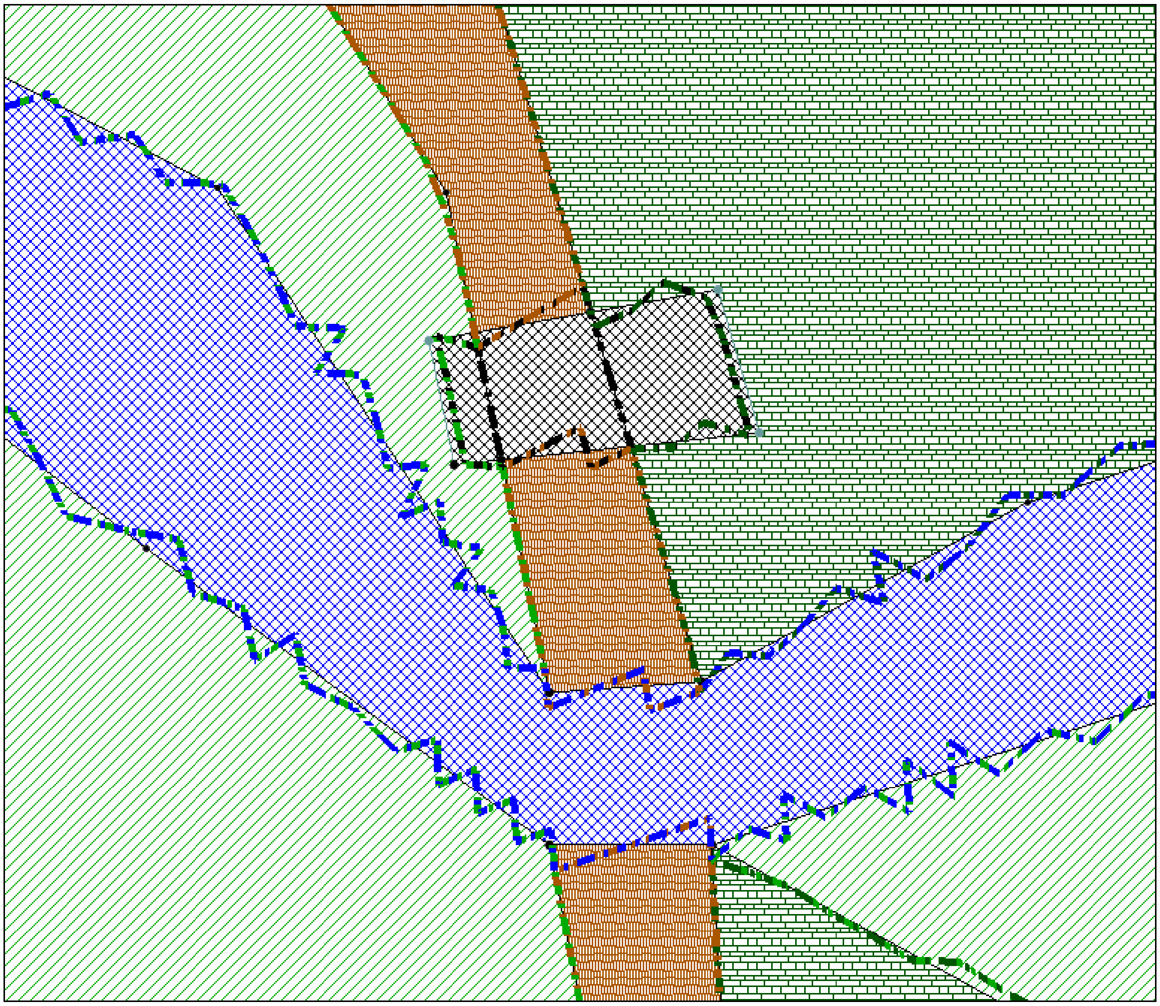


Figure 7 Edited Material Coverage

4 Specifying the Initial Conditions of the Stream

Most of the model control options have been specified and the model is nearly ready to run. One option that still needs to be specified from within the model control is the initial condition of the model. SRH-2D has several options to specify the initial condition of the elements in the model domain. They are as follows:

- **Dry** – All elements in the mesh start out dry.
- **Initial Water Surface Elevation** – Sets the WSE for all elements to one value. If the elevation at an element is greater than the specified WSE, then the element is dry.
- **Automatic** – Assumes backwater from the downstream boundaries with areas above this value assumed dry.

- **Restart file** – Initial conditions defined from a previous solution file. When a simulation is run, SRH-2D will write out a Restart file for each output interval as specified in the *Model Control* under the *Output* tab.

For this model, a restart file will be used. The restart file represents an initial condition equivalent to the normal base flow of the channel.

1. Right-click on the "Culvert Flow" simulation and select **Model Control…** to bring up the *Model Control* dialog.
2. In the dialog, rename the Case Name to "Culvert_Flow".
3. Under *Initial Condition*, select "Restart File" from the combo box.
4. Choose **Select** under *Restart Conditions File*.
5. Navigate to the "Data Files" folder for this tutorial and select "Baseflow_RST25.dat".
6. Choose **Open** to close the file browser.
7. Click **OK** to close the *Model Control* dialog.
8. Select the **Frame** tool to frame the model domain extents.
9. In the Project Explorer check the boxes next to "Mesh Data" and "GIS Data" to turn on the display of the mesh and background image.
10. Turn off the display of the materials coverage by unchecking the box next to the "Culvert Materials" coverage.
11. Select *Display* | **Display Options…** to open the *Display Options* dialog.
12. In the *2D Mesh* section, uncheck the box next to *Elements* to turn off the display of mesh elements. Select **OK** to exit the *Display Options* window.

5 Saving and Running the Simulation

Now that the culvert structure has been created and defined, the model is ready to run.

1. Now would be a good time to save the project. Select *File* | **Save as…**
2. Save the project as "Gila_Culvert.sms".
3. Right-click on the "Culvert Flow" simulation and choose **Save, Export, and Launch SRH-2D**.
4. Select **OK** if a warning is displayed stating that the "Culvert Materials" coverage will be renumbered before exporting.

When saving, exporting and launching SRH-2D, SMS will initialize and run pre-SRH, the SRH-2D preprocessor. When pre-SRH has finished running, SRH-2D will begin to run.

5. It may take 10 minutes or more for the model to run to completion. The program will terminate with a message stating that the "Program terminated with exit code 0, Exit Window?". Select **Yes**.

A set of completed files have been included with the tutorial files located in the "Output" folder. If desired, open a new instance of SMS while the model is running and load in the "Gila_Culvert.sms" SMS project to view the completed model with the results.

6. As shown in Figure 8 make sure *Load Solution* is checked in the SMS model wrapper and click **Exit**. The solution datasets will now be listed in the Project Explorer under "Gila_Mesh".

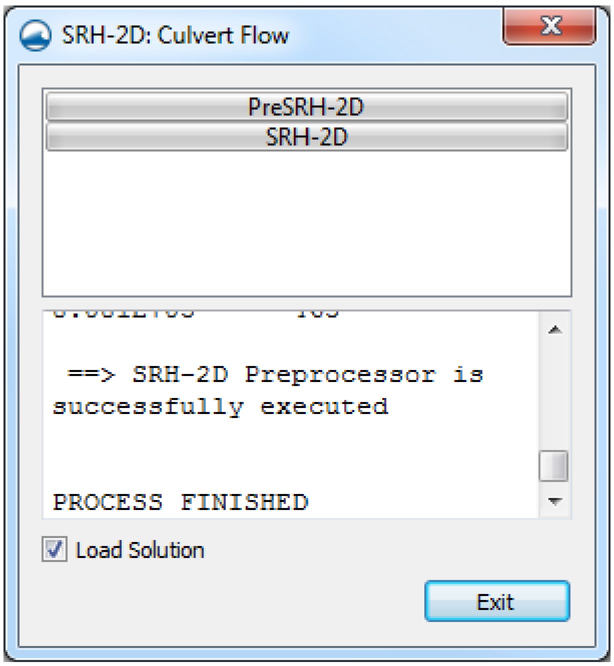


Figure 8. SMS Model Wrapper

5.1 Organizing the Solution Datasets

For better dataset organization, a folder will be created in which the culvert solution datasets may be stored.

1. Right-click on "Gila_Mesh" and select **New Folder**.
2. Rename the new folder as "Culvert Flow".
3. Select the 6 mesh datasets that correspond to the culvert solution by holding down the *Shift* key and selecting the datasets. The datasets are: "Froude", "Strs_lb_p_ft2", "Vel_Mag_ft_p_s", "Velocity", 'Water_Depth_ft", and "Water_Elev_ft".

4. Drag the selected datasets below the "Culvert Flow" folder that was created in steps 1 and 2. The datasets should be organized as shown in Figure 9.

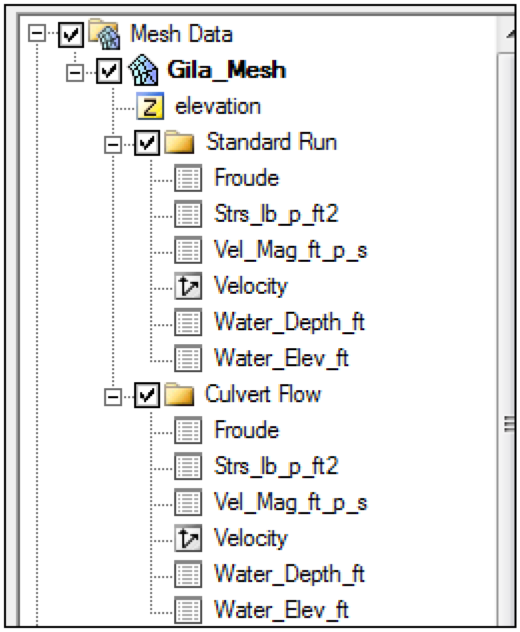


Figure 9 Mesh Dataset Organization

5. Select the "Water_Elev_ft" mesh dataset within the "Culvert Flow" folder to make it the active dataset for viewing.

6 Visualizing the Results

SMS has several ways by which results can be visualized. One useful way to compare the effects of the culvert on the channel is to create a new mesh dataset representing the differences in water surface elevations between the culvert solution and the existing condition solution. The difference dataset can be created using the *Data Calculator*.

1. Toggle through the datasets and time steps to see the results.
2. Toggling through the solution below the "Standard Run" folder, you may notice that water flows over the road around time 00:18:00. Comparing this with the solution created from the culvert model run, you will notice that the water no longer flows across the road.
3. Select *Data* | **Data Calculator…** to bring up the *Dataset Toolbox* dialog.

An expression will be created in the calculator that uses all time steps and takes the difference between the existing condition "Water_Elev_ft" and the culvert "Water_Elev_ft".

4. Click on the "d6. Water_Elev_ft" dataset under the "Standard Run" folder to select and make it active..

5. Check the box to *Use all time steps*.
6. Select the **Add to Expression** button to add the "d6. Water_Elev_ft" dataset to the expression.
7. Select the subtract 

button.
8. Select the "d11. Water_Elev_ft" dataset under the "Culvert Flow" folder and press the **Add to Expression** button to add it to the expression. The expression should now look like the following expression: "d6:all-d11:all".
9. Specify the name of the dataset as "WSE_Diff" in the *Output dataset name* box. The window should look like Figure 10.
10. Select **Compute** to create the new dataset.
11. Select **Done** to close the *Dataset Toolbox*.

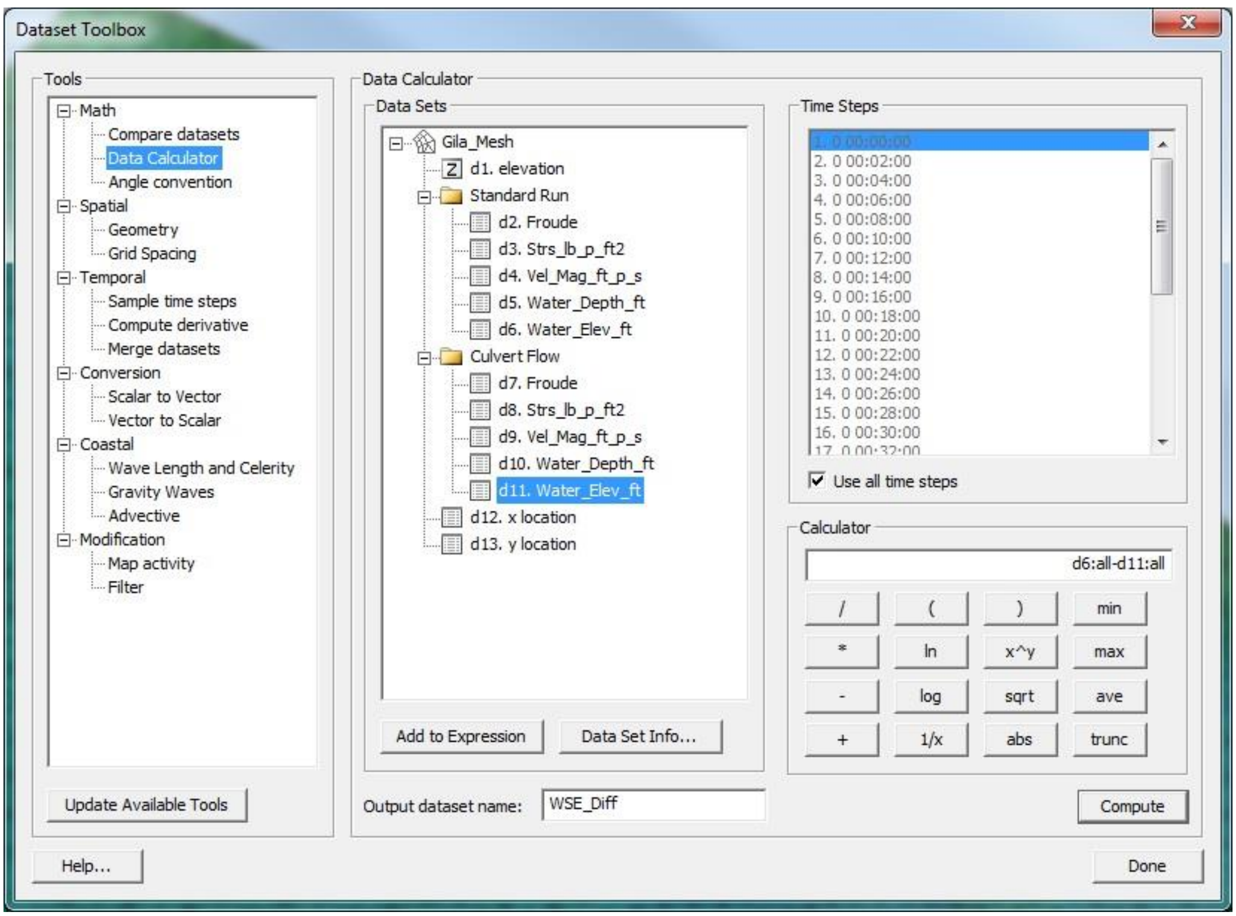


Figure 10 Data Calculator Expression

12. Select the "WSE_Diff" dataset and toggle through the time steps. The positive values represent water surface elevations that were higher in the existing condition

solution and negative values represent water surface elevations that were higher in the culvert solution. Across the domain, the reduction in WSE is evident with the largest differences being located near the culvert structure faces, as expected.

When SRH-2D was run, an output file was created for the culvert structure that includes diagnostic information for the culvert. This file can be a useful way to understand what is happening within the culvert structure. It can be found within the output file directory and is called “Culvert_Flow_CULV1.dat”. It can be opened in a text editor for viewing the flows through the culvert and water surface elevations at the faces of the structure.

7 Conclusion

This concludes the “SRH-2D – Culvert Structures” tutorial. If desired, further analysis could be performed on the solution to evaluate other possible effects of the culvert on the channel.

Topics covered in this tutorial include:

- Opening an existing SRH-2D project
- Creating a culvert boundary condition
- Using a restart file for the initial condition
- Saving and running SRH-2D
- Organizing mesh datasets into folders
- Visualizing and comparing solution results

If desired, continue to experiment with the SMS interface or quit the program.

