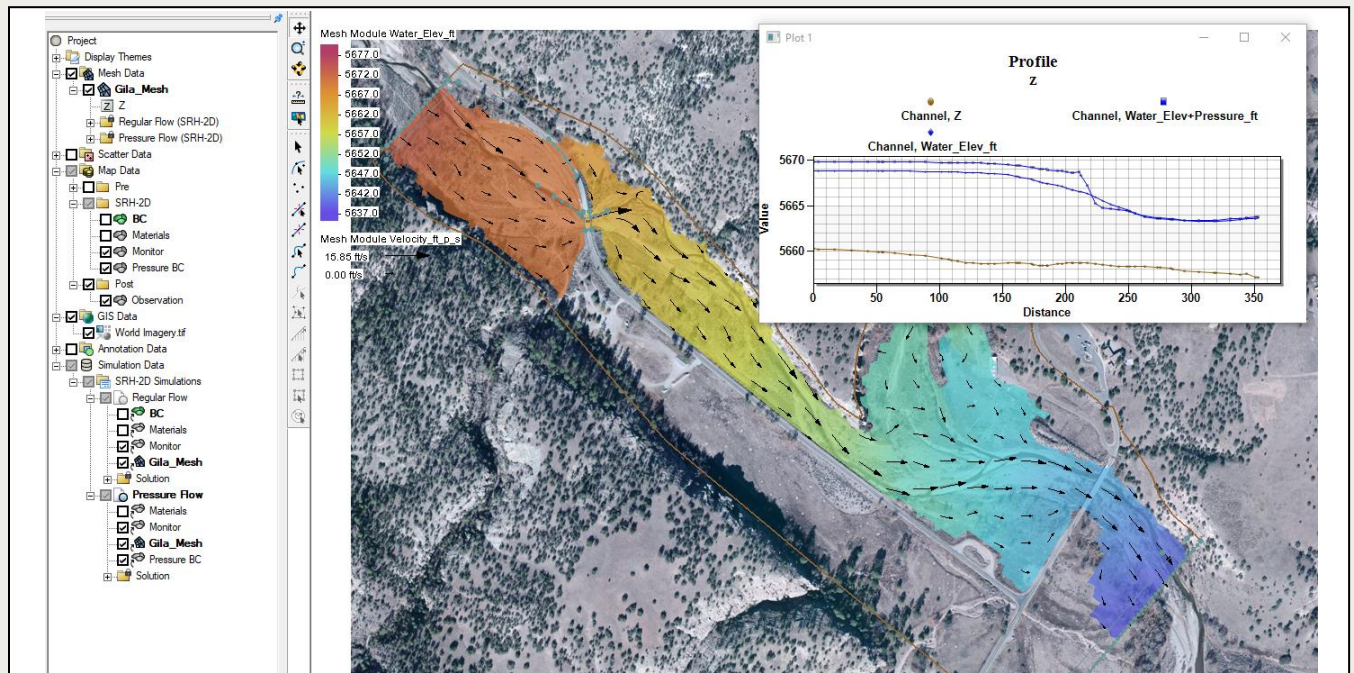


SRH-2D Tutorial

Bridge Pressure Flow

Add bridge pressure flow structures to SRH-2D simulations



Objectives

This tutorial demonstrates the process of creating a pressure flow structure within SRH-2D to model pressurized flow beneath a bridge.

Prerequisite Tutorials

- SRH-2D
- SRH-2D – Simulations

Required Components

- SMS Core
- SRH-2D Model & Interface

Time

- 15–20 minutes

1	Introduction	2
2	Getting Started	2
3	Creating the Pressure Flow Structure	3
3.1	Creating the Structure Arcs	3
3.2	Examining the Mesh Around the Bridge	5
3.3	Assigning the Bridge Deck (Pressure) Attributes	6
4	Saving and Running the Simulation	7
5	Visualizing Results.....	7
5.1	Comparing the Solutions	7
5.2	Using Observation Arcs	7
5.3	Checking Overtopping	9
6	Conclusion.....	11

1 Introduction

This tutorial demonstrates how to create a pressure flow structure within SRH-2D to model pressurized flow beneath a bridge. The SRH-2D “Simulations” tutorial should have been completed before attempting this tutorial. All files referenced in the instructions are found in the “Input” folder within the “SMS_SRH-2D_Pressure” folder.

An existing SRH-2D model will be used to facilitate the setup for this tutorial. The area being modeled is located at the confluence of the west and middle forks of the Gila River in New Mexico.

In this tutorial, an existing bridge upstream of the confluence will be analyzed. One concern with the design is that the elevation of the low chord will impede the flow of the river during high flows. The bridge deck will be represented as a pressure flow structure in SMS. After running the model with the pressure flow bridge, the solution will be compared with an existing condition solution provided to evaluate the effects.

(Note: Pressure zone boundary conditions are limited to planar or parabolic shape. An alternative is to use a 3D bridge to define an arbitrary ceiling elevation as illustrated in the SMS_SRH-2D Pressure Ceiling tutorial. These two methods are mutually exclusive. Only one can be applied in a single SRH-2D simulation.)

2 Getting Started

To begin, do the following:

1. Open a new instance of SMS.
2. Select *File* | **Open...**
3. Navigate to the “Gila_Structure.sms” project found in the *Input* folder for this tutorial.
4. Click **Open** to import the project.

The project should appear similar to Figure 1.

In the Project Explorer, duplicates of the “ Regular Flow” simulation and the “ BC” coverage have been made to facilitate the model setup. The duplicates have been renamed as “ Pressure Flow” and “ Pressure BC” respectively.

The process of duplicating and linking these items to a simulation was demonstrated in the “SRH-2D Simulations” tutorial. Creating duplicates of simulations or coverages allows 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.

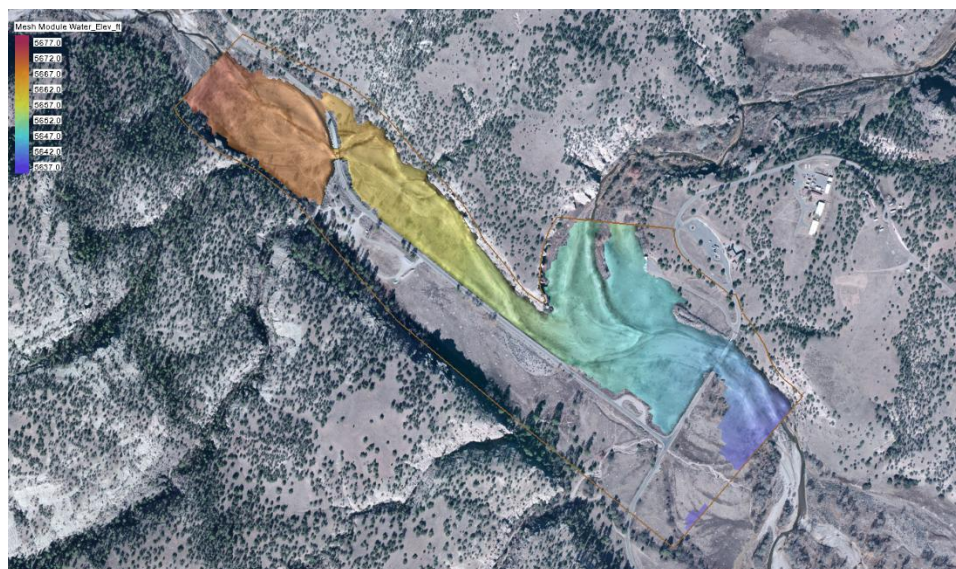


Figure 1 Gila_Structure.sms project

The mesh datasets located under the “Regular Flow (SRH-2D)” folder in the Project Explorer are from an SRH-2D solution of the existing flow conditions, without the pressure flow structure. These results will be used to make comparisons and visualize the effects that the pressure flow structure will have on the model.

3 Creating the Pressure Flow Structure

The pressure flow structure will be created at the bridge (displayed in Figure 2). Pressure flow structures are defined by creating an arc at the upstream side of the bridge deck and an arc at the downstream side of the bridge deck representing the upstream and downstream extents of the bridge. The arc type for these arcs is defined as “Pressure” and attributes are assigned to them such as low chord elevations, a Manning’s n value, and overtopping parameters.

3.1 Creating the Structure Arcs

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

1. Select **Display | Display Options...** to open the *Display Options* dialog.
2. In the *2D Mesh* section, check the box next to *Elements* to turn on the display of mesh elements.
3. Check the box next to *Vectors* to turn on the display of vector arrows.
4. Select **OK** to exit the *Display Options* dialog.
5. Select the “Z” dataset under “Gila_Mesh” to make it active.
6. Use the **Zoom** tool to zoom into the bridge location (Figure 2).

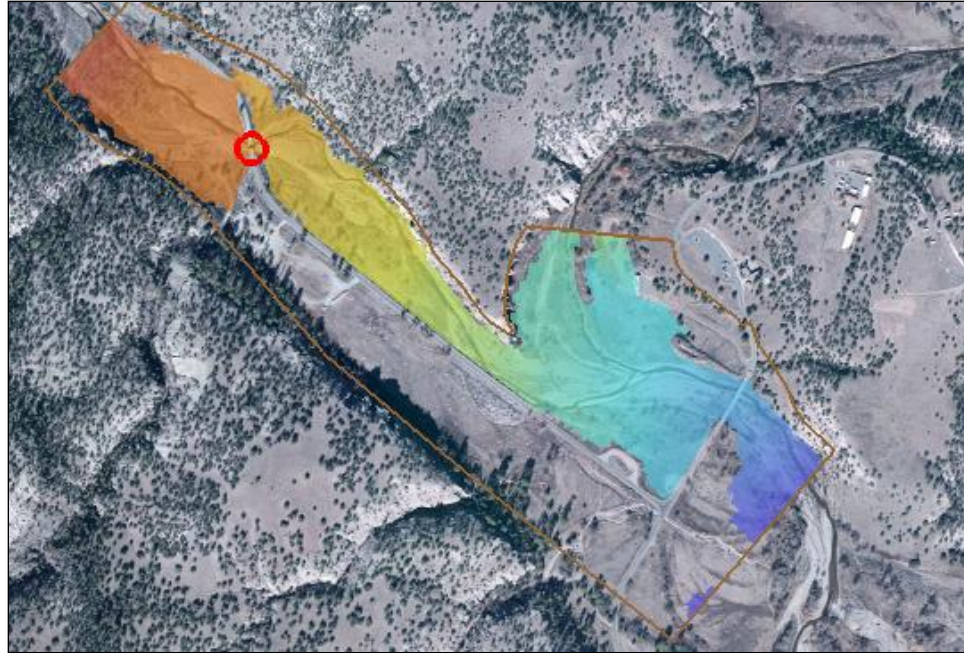


Figure 2 Bridge pressure flow location

7. In the "SRH-2D" folder, select the "Pressure BC" coverage to make it active. Make sure the checkbox is checked to display this coverage.
8. Use the **Create Feature Arc** tool to create one arc on each side of the bridge to define the upstream and downstream faces of the bridge deck. The arcs should be created from right to left or bottom to top. The arcs should look similar to Figure 3.

It is recommended that the mesh be created to contain quadrilateral elements within the area between these two arcs which represent the bridge deck and that the structure arcs are aligned with a clean row of element edges. The arcs should also extend into the embankments.

Note: When drawing these arcs, they should be drawn in the same direction. After the first arc has been drawn, ensure that the second arc is drawn in the same direction (north to south or south to north). Drawing them in opposing directions may cause an error when running SRH-2D.

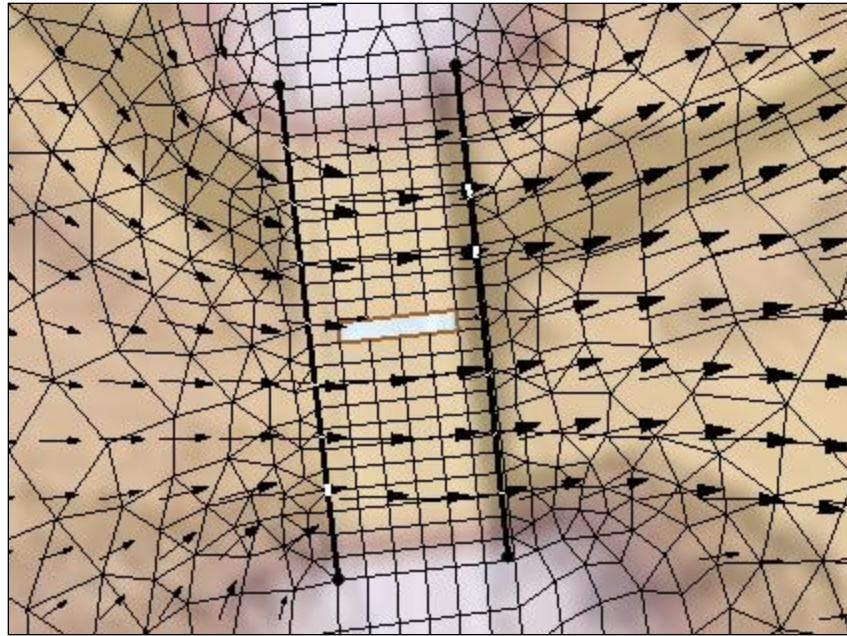


Figure 3 Pressure flow arc locations

3.2 Examining the Mesh Around the Bridge

Care should be taken to ensure that the mesh and the bridge deck are compatible. In other words, they should fit together in space.

To check this, elevations and distances can be measured in the area where the culvert will be inserted into the mesh. Another means of checking this visually is to add a 3D bridge to the project for this location. Refer to the 3D bridge and ceiling elevation tutorials for more information on this feature.

Bridge Dimensions

1. Use the **Select Feature Arc**  tool to select one of the arcs which define the pressures zone.

SMS displays the length in the information window at the bottom of the Graphics Window. The length should be approximately 105 feet. The exact length depends on how the arc was digitized.

Bridge Elevations

The “bridge crest elevation” is the elevation at which water will start overtopping the road. The “ceiling elevations” are the bottom elevations of the bridge structure (up and down stream ends).

To evaluate the elevations:

1. Track the cursor over the roadway at the ends of the embankments and note the elevation by looking at the information window at the bottom of the Graphics Window.

The elevations of the roadway are approximately 5672.0 to 5672.5 feet. A pressure zone simulates a flat weir and a flat or parabolic bridge deck. If there is significant

variation, the pressure ceiling may be a better option (refer to that tutorial). Remember that the crest must assume flowing over guide rails or barriers on top of the bridge unless the assumption is made that this portion of the structure will be washed away. So, the crest elevation in this situation could be approximated at 5673.5 feet.

The ceiling elevation is based on the thickness of the structure measured down from the crest elevation. In this case, assume the structure is 8.5 feet thick. Therefore, the ceiling elevation would be 5665.0 feet.

If this bridge is to be constructed (design mode), the mesh elevations could be modified along with the bridge thickness to match the anticipated construction. If this simulation represents existing conditions, care should be taken to make sure the mesh represents the conditions being simulated.

3.3 Assigning the Bridge Deck (Pressure) Attributes

When the arcs for the bridge deck were created, SMS defaults them to be “Walls (no-slip boundary)” for SRH-2D. This is the default boundary condition type. Therefore, the next step in creating the structure is to specify the boundary condition type and define the pressure attributes.

1. Using the **Select Feature Arc**  tool, select the upstream (leftmost) arc. Take note of the ID for this arc which is displayed in the help window below the SMS display window.
2. While pressing the *Shift* key, select the downstream arc so that both of the arcs are selected.
3. Right-click on either arc then select **Assign BC...** to bring up the *SRH2D Assign BC* dialog.
4. In the *BC Type* drop-down, select “Pressure”.

Note the assignment of “Upstream” and “Downstream” to the two arcs, associated with their ID values. If the ID displayed for pressure upstream is not the same as noted above in step 1, switch the associations using the *Arc id* drop-downs.

5. In the *Pressure options* section, the *Units* should be left as “Feet”.
6. Enter “5665.0” for both *Ceiling elevation along upstream (Flat) or edges (Parabolic)* and *Ceiling elevation along downstream (Flat) or midspan (Parabolic)*.

These ceiling elevations represent the low chord for the bridge and can be different for the upstream and downstream faces of the bridge deck.

7. Enter “0.09” for the *Manning roughness coefficient between water and ceiling*.
8. Turn on *Model overtopping on this zone*.
9. Enter “5673.5” as the *Crest elevation* for overtopping. This corresponds to the road surface in this case.
10. Enter “103.0” as the *Length of weir over pressure zone*. This is the lateral length of the portion of the bridge which will undergo weir flow if overtopped.
11. Click **OK** to exit the *SRH2D Assign BC* dialog.

Note: The *General structure options* will not be used in this example. If there is high skew or rapid drawdown at the entry or exit of the structure, then consider using a [BC Data line](#).

4 Saving and Running the Simulation

Now that the pressure flow structure has been created, the model is ready to run.

1. Right-click on the “ Pressure Flow” simulation and select **Save Simulation**.
2. After the project has saved, right-click on “ Pressure Flow” simulation and select **Run Simulation**.
3. Click the **Load Solution** button in the *Simulation Run Queue* dialog to import the solution file.
4. Click **Close** to exit the *Simulation Run Queue*.

5 Visualizing Results

With the solution datasets read into the SMS project, create a 2D plot of the water surface elevations to compare the pressure flow solution with the original solution without pressure flow.

5.1 Comparing the Solutions

The completed pressure flow simulation invites the comparison of the two solutions. To do this:

1. Select *Display* | **Display Options...** to open the *Display Options* dialog.
2. In the *2D Mesh* section of the dialog, uncheck the box next to *Elements* to turn off the display of mesh elements.
3. Click **OK** to close the *Display Options* dialog.
4. Expand both simulation folders in the Project Explorer and select the “ Water_Elev_ft” dataset within the “ Pressure Flow (SRH-2D)” folder.
5. Use the **Zoom**  tool to zoom into the area of the bridge and the section of roadway that is being flooded to the left of the bridge.
6. Switch back and forth between the “ Water_Elev_ft” dataset in the two solution folders.

Note the increased width of the overtopping region and the higher water level.

5.2 Using Observation Arcs

Observation arcs and associated plots can help visualize and quantify the difference in the solutions. The initial project includes an observation coverage with two arcs. One arc follows the road in the area of the overtopping flow. The other follows the channel through the bridge opening. To review how to create these arcs refer to the “Map Module” tutorial.

The following sections will use these arcs to measure the flow over the road and visualize the difference between the water levels.

Computing the Overtopping Flow

To compute the difference in overtopping flow:

1. Under the “ Pressure Flow (SRH-2D)” folder, select the “ Water_Elev_ft” dataset make it the active dataset.
2. Under the “ Post” folder, select the “ Observation” coverage.
3. Using the **Select Feature Arc**  tool, double-click on the arc along the road in the overtopping flow to bring up the *Observation Coverage* dialog with the “Road” arc selected, as shown in Figure 4.
4. In the *Flux Measurements* section at the top of the dialog, enter the name “Flow” and press the *Tab* key.

This will create a new measurement entity and default the attributes of the measurement to the 2D mesh module and the “Z” and “Velocity” datasets.

5. Change the *Scalar Data Set* to “Water_Depth_ft”.

Notice in the bottom section of the dialog that the “Road” arc now has a *Computed Value* of “416.574” cfs. This represents the simulated overtopping flow rate for the pressure flow simulation.

6. Click **OK** to close the *Observation Coverage* dialog.
7. In the Project Explorer, select the “ Water_Elev_ft” dataset within the “ Regular Flow (SRH-2D)” folder to make it the active dataset.
8. Reselect the “ Observation” coverage
9. Use the **Select Feature Arc**  tool and double-click on the arc along the road in the overtopping flow again.

The *Observation Coverage* dialog opens again, but now the *Computed Value* of “28.148” cfs represents the simulated overtopping flow rate for the regular flow simulation.

10. Click **OK** to close the *Observation Coverage* dialog.



Figure 4 Selected arc along the road

Viewing a Profile Plot

An observation profile plot provides another way to compare solutions. To create a 2D plot of the water surface elevation profiles:

1. Use the **Select Feature Arc**  tool and select the channel arc.
2. Right-click and select **Show Observation Plot** to open an observation profile plot.
3. Right-click on the plot and select **Plot Data...** to bring up the *Data Options* dialog.
4. Choose *Specified* under *Dataset(s)*.
5. Check the boxes next to the  "Water_Elev_ft" dataset under both the  "Regular Flow (SRH-2D)" folder and under the  "Pressure Flow (SRH-2D)" folder.
6. Check the box next to the  "Z" dataset.
7. Select **OK** to close the *Data Options* dialog.
8. Right-click on the plot again and select *Legend | Top* to turn on the legend.

The effects of the bridge are apparent as the water is backed up and forced below the bridge deck.

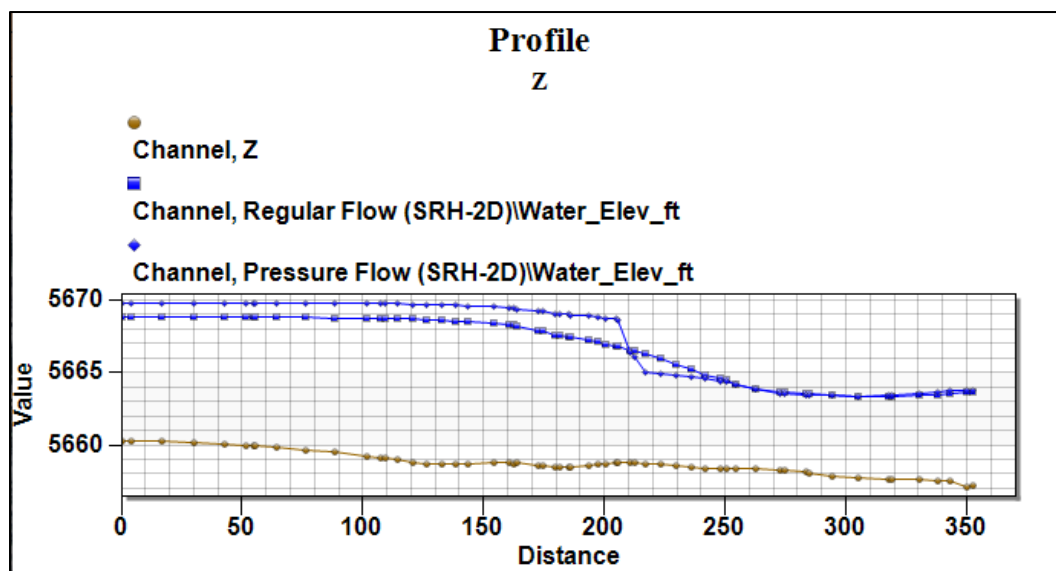


Figure 5 Water Surface Elevation Profile Plot

9. Minimize the plot to get it out of the way.

5.3 Checking Overtopping

This case was run with overtopping allowed across the pressure flow structure. A crest elevation of 5673.5 feet was specified in section 3.3 corresponding to the elevation of top of the roadway. For pressure flow structures, only the flow going over the roadway in the area of the bridge deck is calculated. Overtopping of the roadway in other areas occurs over the terrain within the 2D model domain.

The overtopping flow is reported in a separate output file in the simulation results directory. This file can be reviewed as a structure plot or manually opening the file. To review the information in a plot:

1. Select the “ Pressure BC” coverage to make it active.
2. Use the **Select Feature Arc**  tool and select one of the pressure arcs in the “ Pressure BC” coverage.
3. Right-click on the arc and select **Structure Output Plots...** to open the *SRH-2D Solution Plots* dialog.

The *Discharge* plot is defaulted and shows a curve at 0.0 indicating no flow overtops the bridge during this simulation.

4. Turn off the box next to *Discharge(cfs)* and turn on both the *WSE_up(ft)* and the *WSE_down(ft)* plots.

These plots show the water level at the upstream and downstream faces of the pressure zone that are used to compute the flow over the bridge deck using the weir equation. Sometimes SRH-2D will output initial values that are nonsensical. In this case, an initial downstream water level of $-3.0e10$ appears. The next steps will mask out this bogus value.

5. Under the X axis section, turn on the *Specify range* option.
6. Enter a *Minimum time* of “0.5” hours and click the *Tab* key.

The plot now shows an upstream water level that converges during the simulation to a value around 5668.5 feet. This is not high enough to generate overtopping flow.

7. Click **Close** to close the *SRH-2D Solution Plots* dialog.
8. Outside of SMS, navigate to the simulation results directory. In this case, this will be the `.../SMS_SRH-2D_Pressure /Input/Gila_Structure/SRH-2D/Structure Flow` directory, which has the results for the current *Structure Flow* simulation.

The *Case Name* for our solution is “Pressure_Flow”.

9. Navigate into the *Output_MISC* folder and open the file “Pressure_Flow_INTERNAL1.dat” in a text editor.

Note: The output file for the structure can be found in `\Gila_Structure\SRH-2D\Pressure Flow\Output_MISC\Pressure Zone_Run_INTERNAL1.dat`. This is a text file that shows the computed values for every 100 time steps. In this case, that is 200 seconds or 0.05556 hours.

6 Conclusion

This concludes the “SRH-2D Bridge Pressure Flow”¹ tutorial. Further analysis could be performed on this solution to evaluate other effects of the bridge on the channel. The topics demonstrated in this tutorial include:

- Opening an existing SRH-2D project.
- Creating a pressure flow structure at a bridge location.
- Saving and running SRH-2D.
- Computing flux across an observation arc.
- Creating an observation profile plot.
- Using an observation arc to create a water surface elevation profile plot to visualize results.
- Generating a structure plot to review bridge overtopping.

You may want to continue to experiment with this simulation by doing some of the following:

- Vary the bridge ceiling elevation.
- Vary the crest elevation for overtopping the bridge.
- Change the Manning’s N value specified for the underside of the bridge.

Try changing the bridge to a 3D bridge.

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