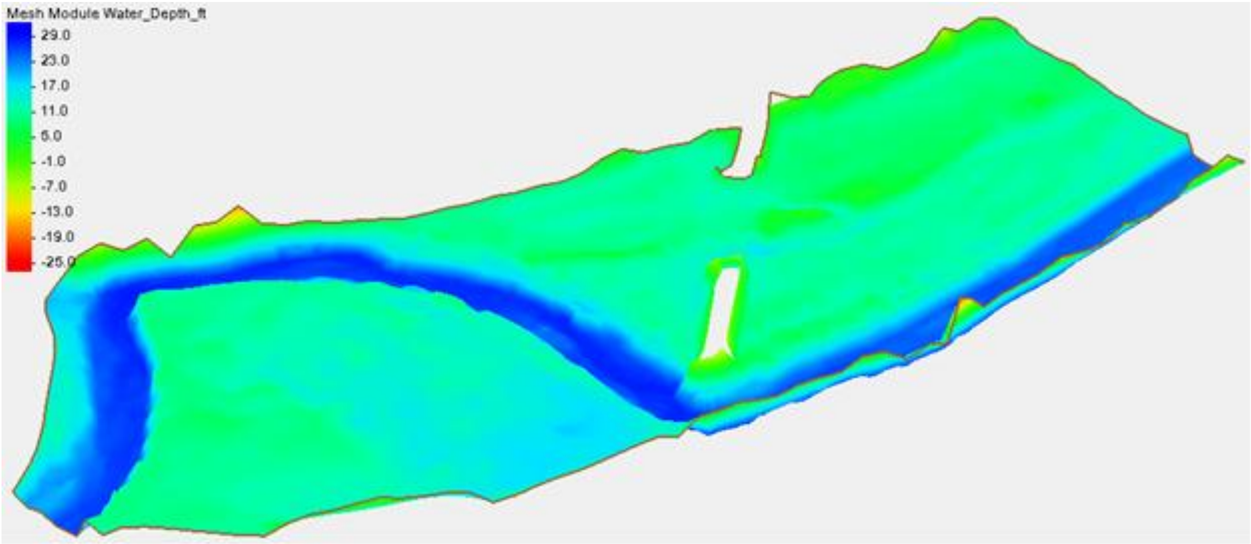
SMS 14.0 Tutorial

SRH-2D

Introduction to creating an SRH-2D model in SMS



Objectives

This tutorial shows how to build a Sedimentation and River Hydraulics – Two-Dimensional (SRH-2D) simulation.

Prerequisite Tutorials

- Overview

Required Components

- SMS Core
- SRH-2D Model & Interface

Time

- 20–30 minutes

AQUAVEO™

1 Introduction

The Sedimentation and River Hydraulics – Two-Dimensional (SRH-2D) model is a two-dimensional (2D) hydraulic, sediment, temperature, and vegetation model for river systems developed at the Unites States Bureau of Reclamation (USBR) and sponsored by the United States Federal Highway Administration (FHWA).

SRH-2D uses a flexible mesh that may contain arbitrarily shaped cells. A hybrid mesh of quadrilateral and triangular cells is used. SRH-2D employs very robust and stable numerical schemes with a seamless wetting-drying algorithm.

SRH-2D is particularly useful for problems where 2D effects are important. Examples include flows with in-stream structures, through bends, perched rivers, side channel and agricultural returns, and braided channel systems. SRH-2D can be run using U.S. customary or metric units.

2 Getting Started

In SMS, a projection should be defined for all projects. Background information, such as topographic, bathymetric surveys, and land use data provides the basis for any hydraulic simulation. All of these have been included in a project file.

1. Launch the SMS application.
2. Select *File* | **Open…** to bring up the *Open* dialog.
3. Browse to the *data files* folder for this project.
4. Select “CimarronTutorial.sms” and click **Open** to import the project and exit the *Open* dialog.

The project contains an unstructured grid (or mesh) as well as the data used to generate this mesh. For information on building a mesh, refer to one of the mesh generation tutorials. The display of SMS should be like *Figure 1*. The project contains an online background map that should appear if there is internet access.

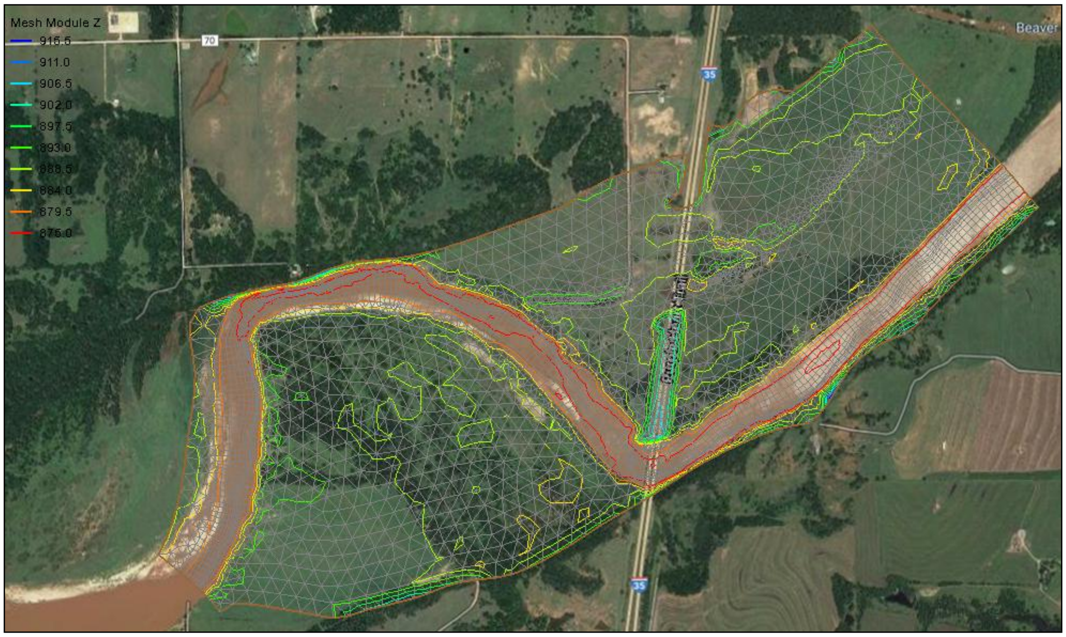


Figure 1 The initial project

3 Assigning Boundary Conditions

Boundary conditions force hydrodynamic conditions into the model. For this model, an inflow boundary condition will be specified at the upstream boundary, and water surface elevation at the downstream end of the model.

3.1 Creating the Boundary Condition Coverage

In SRH-2D, the boundary conditions come from a SRH-2D boundary condition coverage. This coverage will become part of the SRH-2D simulation. To create the boundary conditions coverage for this simulation:

1. Turn off " GIS Data" in the Project Explorer.
2. Right-click on " Map Data" in the Project Explorer module and select **New Folder**. Change the name of the folder to "SRH-2D". (Organizing coverages in folders is optional. This tutorial demonstrates one method that can be used.)
3. Right-click on " Map Data" in the Project Explorer module and select **New Coverage** to open the *New Coverage* dialog.
4. Under *Coverage Type*, select *SRH-2D* | **Boundary Conditions**.
5. Enter "50 yr" as the *Coverage Name* and click **OK** to close the dialog.

This creates the " 50 yr" coverage that is used to define boundary conditions.

6. Drag the " 50 yr" coverage into the "SRH-2D" folder. This also makes it active.
7. Using the **Create Feature Arc** tool, create an arc near the inflow boundary as shown in Figure 2.

The arc can be inside or outside of the domain, but it is normally less cluttered if it is just outside. The arc properties will be "snapped" to the mesh domain during the model run.

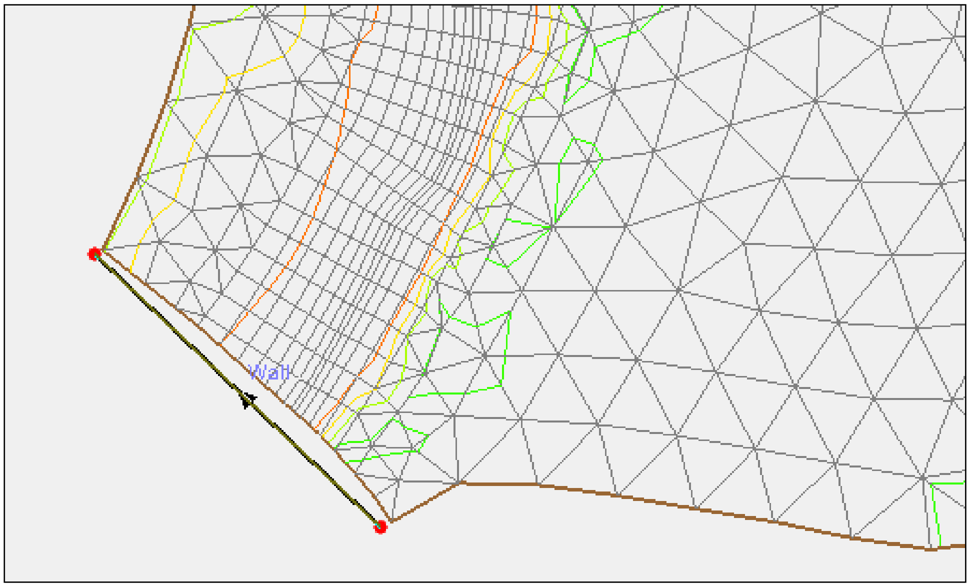


Figure 2 New inflow arc

8. Repeat step 6 to create an arc just outside the domain on the downstream end as shown in Figure 3.

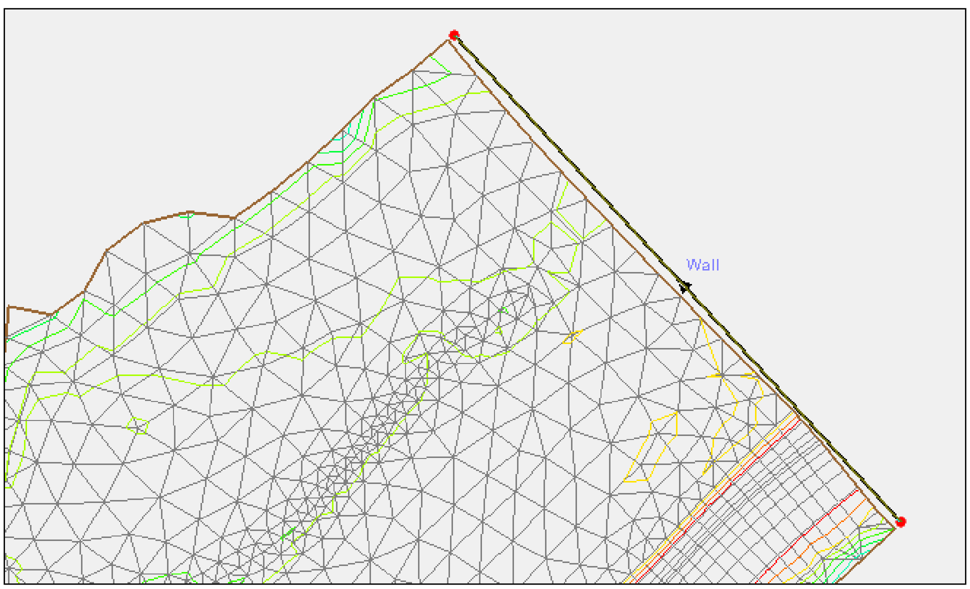


Figure 3 Upstream and downstream arcs selected

3.2 Assigning the Inflow Boundary Condition

1. Using the **Select Feature Arc** tool, double-click on the arc on the left (upstream) side to open the *SRH2D Assign BC* dialog.
2. In the *BC Type* section, select “Inlet-Q (subcritical inflow)” from the *BC Type* drop-down.
3. Select “Constant” from the *Discharge option* drop-down.

4. Make sure the units for the *Constant Q* drop-down is set to “cfs” and enter a value of “34750.0”.
5. For the *Label*, enter “Inlet-Q=34750.0”.
6. Click **OK** to exit the *SRH-2D Linear BC* dialog.

3.3 Assigning the Outflow Boundary Condition

1. Using the **Select Feature Arc** tool, double-click on the arc on the right (downstream) side of the coverage to open the *SRH2D Assign BC* dialog.
2. In the *BC Type* section, select “Exit-H (subcritical outflow)” from the *BC Type* drop-down.
3. Select “Constant” from the *Water surface (WSE)* drop-down.
4. Ensure the units are specified as “Feet” from the *Constant WSE* drop-down and enter “896.0”. (The populate button computes an approximate “normal depth” elevation-based Manning’s equation. This is demonstrated in another tutorial.)
5. For the *Label* enter “WSE=896.0”.
6. Click **OK** to exit the *SRH2D Assign BC* dialog.
7. Open **File** and select *Save as…*then save the project under a new file name.

4 Assigning Material Properties

In nature, the vegetation and shape of each element of the mesh resists flow of water through that area. For SRH-2D, this resistance is specified as a material type with an associated Manning’s *n* roughness coefficient.

The material zones are assigned using polygons in the SRH-2D materials coverage.

4.1 Creating the Materials Coverage

To create the materials coverage for this simulation:

1. Right-click on the “ I-35 MeshGen” coverage and select **Duplicate**.
2. Right-click on “ I-35 MeshGen (2)” and select **Rename**.
3. Enter “Materials” and press *Enter* to set the new name.
4. Drag the “ Materials” coverage into the “SRH-2D” folder.
5. Right-click on “ Materials” and select *Type* | *Models* | *SRH-2D* | **Materials**.

The polygons that exist in this coverage that were used for mesh generation already define the principal zones: channel, flood plain, and highway. Therefore, no additional feature generation is required. There are extra features in this coverage. They are not required to define material zones, but they do not have any negative impact so they can be left alone.

This illustrates how geometry (points, arcs, and polygons) can be reused and don’t need to be digitized multiple times.

4.2 Creating the Materials List

The SRH-2D simulation can use as many different material types as assigned in the *Material Properties* dialog. In this simulation, three material types, each with its own roughness characteristics will be defined.

To do this:

1. Right-click on " Materials" and select **Material List and Properties** to bring up the *Material List and Properties* dialog.
2. Click the **Add Material** button three times to create three new material types.
3. Double-click each new material name in turn and enter the name as indicated in the table below.
4. Enter the roughness value for each material in the *Manning's N* field to the right of each selected entry.
5. Specify a color and pattern for each material by clicking the large button to the left of each material name.
6. Click **OK** to close the *Material List and Properties* dialog.

Materials Name	Manning's *n*
Flood Plain	0.045
Road	0.025
River	0.03

The material list has now been defined.

4.3 Assigning Material Types to Material Polygons

Each material polygon must have a material type assigned to define what resistance to flow will exist for elements in that area. To do this:

1. Turn off " Mesh Data" in the Project Explorer.
2. Turn on the " Materials" coverage and select it to make it active.
3. Using the **Select Feature Polygon** tool, double-click on one of the five polygons to bring up the *Assign Material* dialog.
4. Select the appropriate material type for that zone, as shown in Figure 4 below.
5. Click **OK** to close the *Assign Material* dialog.
6. Repeat steps 3–5 for each of the other four polygons.

Note: The material zones can be displayed by opening the *Display Options* dialog by clicking **Display Options** , turning on the *Fill* option under the *Polygon* section of the *Map* tab and making sure it is not 100% transparent. This display can conflict with the elevation's contours, so turning one or the other display option off is helpful for better visualization.

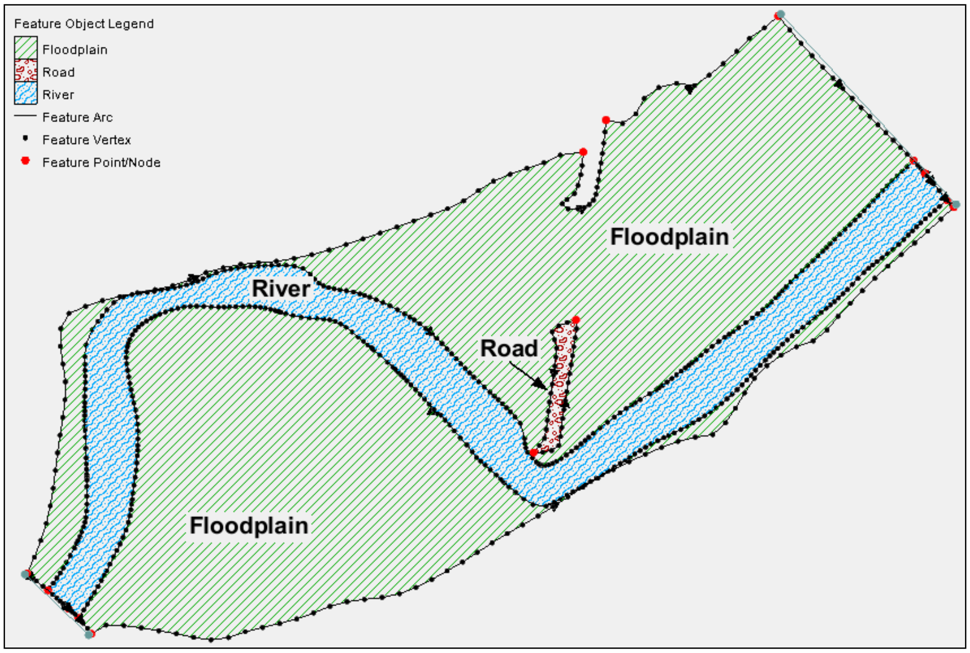


Figure 4 Material types of the four polygons

5 Defining the SRH-2D Simulation

The SRH-2D simulation is created as a separate entity in the SMS project. Individual components for the simulation are associated (or linked) to the simulation and model parameters are assigned to the simulation.

To create the simulation:

1. Right-click in the white space at the bottom of the Project Explorer and select *New Simulation* | **SRH-2D** to create a new " Sim" simulation.
2. Right-click on " Sim" and select **Rename**.
3. Enter "Steady State" and press *Enter* to set the new name.

5.1 Applying Components

The components are added to a simulation by either right-clicking on a component and selecting the *Apply to* command or by dragging and dropping the component into the simulation.

1. Right-click on the " I-35" mesh and select the *Apply to* | **SRH-2D Simulations** → **Steady State** command.
2. Repeat step 1 for the " 50 yr" and " Materials" coverages.

5.2 Assigning SRH-2D Parameters

It is also necessary to set up the SRH-2D control data and define the model parameters by doing the following:

1. Right-click on " Steady State" and select **Model Control…** to bring up the *SRH2D Model Control* dialog.
2. On the *General* tab, enter "Cimarron" as the *Simulation Description* and "Steady" as the *Case Name*. (These have no impact on the simulation. They are for reference only.)
3. Enter "0.0" as the *Start Time (hours)*.
4. Enter "5.0" as the *Time Step (seconds)*.
5. Enter "2.5" as the *End Time (hours)*.
6. Select "Automatic" from the *Initial Condition* drop-down.
7. On the *Output* tab, select "XMDF" from the *Output format* drop-down.
8. Select "English" from the *Output units* drop-down.
9. Select "Simulation End" from the *Output method* drop-down.
10. Click **OK** to close the *SRH2D Model Control* dialog.

6 Running SRH-2D

The simulation is now ready to run. This involves four steps.

- *Exporting the SRH-2D files:* SMS writes out an *SRH-2D* folder in the same directory where the SMS project is saved. The SRH-2D folder has several files which have mesh geometry information, boundary conditions, and material definitions, as well as model control information defined in the model.
- *Running the pre-processor:* SRH-Pre is a pre-processor that imports the files exported from SMS and creates an input file for SRH-2D.
- *Running SRH-2D*: After running SRH-Pre, an input file (*.dat) is written out which is used to run the SRH-2D hydrodynamic engine.
- *Running the post-processor:* After the model runs, a post-processor creates the final output files for SMS to read in.

To invoke this command:

1. Right-click on the " Steady State" simulation and select **Save Project, Simulation and Run** to bring up the *Simulation Run Queue* dialog.

This command saves the SMS project, exports all necessary input files, and then runs three applications related to the SRH-2D engine. These include PreSRH-2D (a pre-processor), SRH-2D, and SRH-Post (a post-processor). SMS renumbers the coverages as needed when exporting the files for SRH-2D.

2. If advised of the renumbering, click **Yes** to close the notice.

While executing the run command, SMS assumes that SRH-2D executables are stored in the SMS installation folder. If it cannot find those executables, a prompt will ask to manually locate them. This is a one-time process, and SMS will remember the location of the executables after this.

As the model run progresses, the *Simulation Run Queue* will display the simulation run which can be expanded to show two status bars indicating progress through the preprocessor and the SRH-2D simulation. The entire model run should take a few minutes, depending on the speed of the computer being used. While the simulation is running, SMS allows work to be done in the interface if desired. When finished, SRH will display a completion message in the *Command Line* section of the *Simulation Run Queue* dialog.

3. Click **Load Solution** to import the solutions into SMS.
4. Click **Close** to exit the *Simulation Run Queue* dialog.

7 Viewing SRH-2D Output

Once the SRH-2D run completes, it will generate a XMDF file which will have a name "Steady_XMDF.h5". SMS loads this file in by default if the option in the model wrapper is not unchecked.

This file includes at least the following datasets:

- " B_Stress_lb_p_ft": bottom stress at the ground in newtons per square feet
- " Froude": dimensionless Froude number (< 1 indicates sub-critical flow)
- " Vel_Mag_ft_p_s": velocity magnitude in feet per second
- " Water_Depth_ft": depth of water in feet
- " Water_Elev_ft": water surface elevation in feet above the data
- " Velocity_ft_p_s": *x* and *y* components of velocity in feet per second

Note: SRH-2D also supports options for additional dataset computation in a variety of situations. Refer to other SRH-2D tutorials to learn about these.

Use the visualization tools in SMS as described in the "Data Visualization" tutorial or other tutorials to evaluate the solution datasets created by the SRH-2D model.

8 Conclusion

This concludes the "SRH-2D" tutorial for SMS. The tutorial demonstrated and discussed the following:

- Loading a mesh for SRH-2D
- Creating SRH-2D boundary conditions
- Setting up an SRH-2D simulation
- Defining SRH-2D model parameters
- Running SRH-2D

Feel free to continue experimenting with the project or exit the program.