SMS 13.1 Tutorial
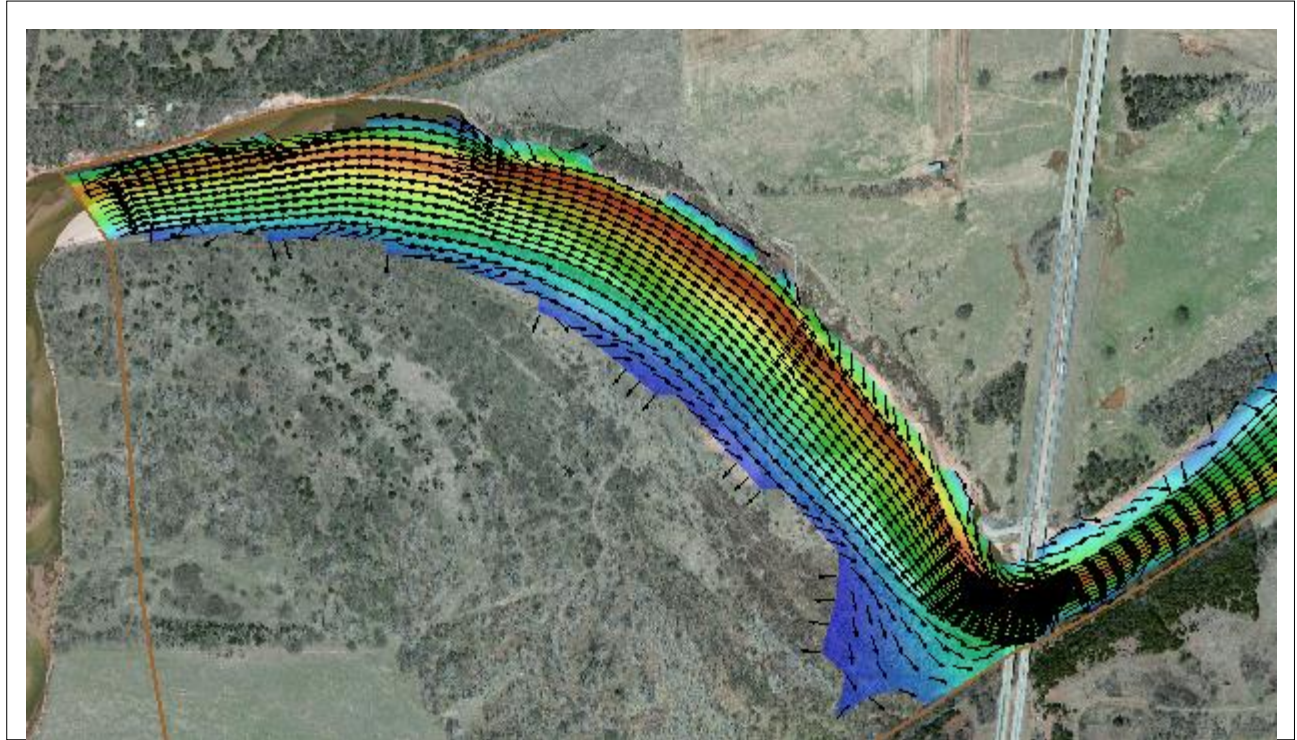
SRH-2D – Post-Processing

Objectives

This tutorial illustrates some techniques for manipulating the solution generated by the Sedimentation and River Hydraulics – Two-Dimensional (SRH-2D) engine using SMS.

Prerequisites

- SMS Overview tutorial
- SRH-2D

Requirements

- SRH-2D Model
- Map Module
- Mesh Module
- Data files

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).

This tutorial builds on previous SRH-2D tutorials and illustrates some techniques for manipulating and visualizing solution data from the engine.

2   Opening the SMS Project

Begin by opening an existing SMS project that includes an SRH-2D simulation with its solution:

1.   Select *File* / **Open** to bring up the *Open* dialog.

2.   Select "Project Files (*.sms)" from the Files of type drop-down.

3.   Browse to the *data files* folder for this tutorial and select "CimarronTutorial.sms".

4.   Click **Open** to open the file into SMS and exit the *Open* dialog.

5.   Select " Water_Depth_m" in the Project Explorer to make it active.

3   Viewing SRH-2D Output

The output files will be imported into SMS as mesh datasets. The Project Explorer may be used to select the desired scalar   and vector   output datasets.

3.1   Creating an Activity Dataset

When the model runs, a large portion of the model domain generally remains dry or unchanged (at least at the initial part of the simulation). When datasets are exported, the dry mesh elements will have depth or velocity of "0.0" and SMS will plot contours in these areas as if there is water everywhere in the domain. For example, see Figure 1 showing flow depth.

Figure 1   Flow depth on a dry mesh

The SMS dataset toolbox has a tool called *Activity dataset* which allows creating a new version of the solution data where the mesh elements values will be exported only if there is any change (activity) in the reference dataset.

In this tutorial, the water surface elevation will be used as the reference data so that the solutions will be exported only if there is any change in water surface elevation at any element at any time step. The elements without change in the activity dataset will be assigned the "no data" value of "-9990.0".

First, create an activity dataset for the velocity magnitude.

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

2. In the *Tools* section, select "Map activity" under the *Modification* item.

3. In the *Value Dataset* section, select "📊 Vel_Mag_m_p_s".

4. In the *Activity Dataset* section, select "📊 Water_Elev_m".

5. Enter "vel_mag_activity" as the *Output dataset name*.

6. Click **Map** to generate the new dataset "📊 vel_mag_activity".

Now to create an activity dataset for the water depth:

7. In the *Value Dataset* section, select "📊 Water_Depth_m".

8. In the *Activity Dataset* section, select "📊 Water_Elev_m".

9. Enter "water_depth_activity" in the *Output dataset name* field.

10. Click **Map** to generate the new dataset "📊 water_depth_activity".

Finally, create an activity dataset for the velocity.

11. In the *Value Dataset* section, select "📊 Velocity".

12. In the *Activity Dataset* section, select "📊 Water_Elev_m".

13. Enter "velocity_activity" as the *Output dataset name*.

14. Click **Map** to generate the new dataset "📊 velocity_activity".

15. Click **Done** to close the *Dataset Toolbox* dialog.

3.2   Scalar Dataset Options

A good way to view the output is to edit the contour display options by doing the following:

1. Select "⊞ water_depth_activity" in the Project Explorer to make it active.

2. Click **Display Options** 🖵 to bring up the *Display Options* dialog.

3. Select "2D Mesh" from the list on the left.

4. On the *2D Mesh* tab, turn off everything except *Contours* and *Mesh boundary*.

5. On the *Contours* tab, in the *Contour method* section, select "Color Fill" from the drop-down.

6. In the *Contour interval* section, select "Number" from the drop-down, and enter "25" in the field to the right of the drop-down.

7. Below the *Contour interval* section, enter "50" as the *Transparency*.

8. Click **OK** to exit the *Display Options* dialog.

9. Turn off "🗀 Map Data" in the Project Explorer.

10. In the *Time step* field below the Project Explorer, select "0 01:00:00" from the list of time steps (Figure 2).

11. Use down arrow key on the keyboard and scroll through the time steps to see how water progresses downstream.
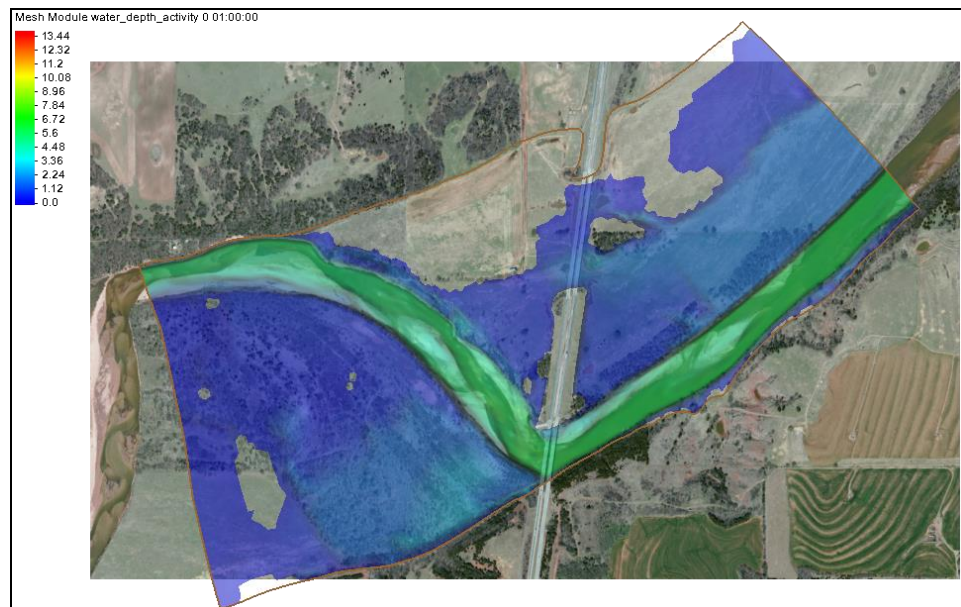

Figure 2   Depth dataset contoured (water_depth_activity)

3.3   Vector Dataset Options

It is possible to display velocity vectors several different ways. This tutorial will first view them displayed at each node, and then on a normalized grid.

To view the vectors at each node:

1.   Select " ⤢ Velocity" in the Project Explorer to make it active.

2.   Click **Display Options** 🖥 to open the *Display Options* dialog.

3.   Select "2D Mesh" from the list on the left.

4.   On the *2D Mesh* tab, turn on *Vectors*.

5.   On the *Vectors* tab, in the *Vector Display Placement and Filter* section, select "at each node/cell" from the *Display* drop-down.

6.   Enter "5.0" as the *Offset*.

The *Z-offset* raises the origin of the vectors so that they are completely visible. Sometimes the vectors can display below the contours, and this option corrects that.

7.   Click **OK** to close the *Display Options* dialog.

8.   Zoom in to the area shown in Figure 3.

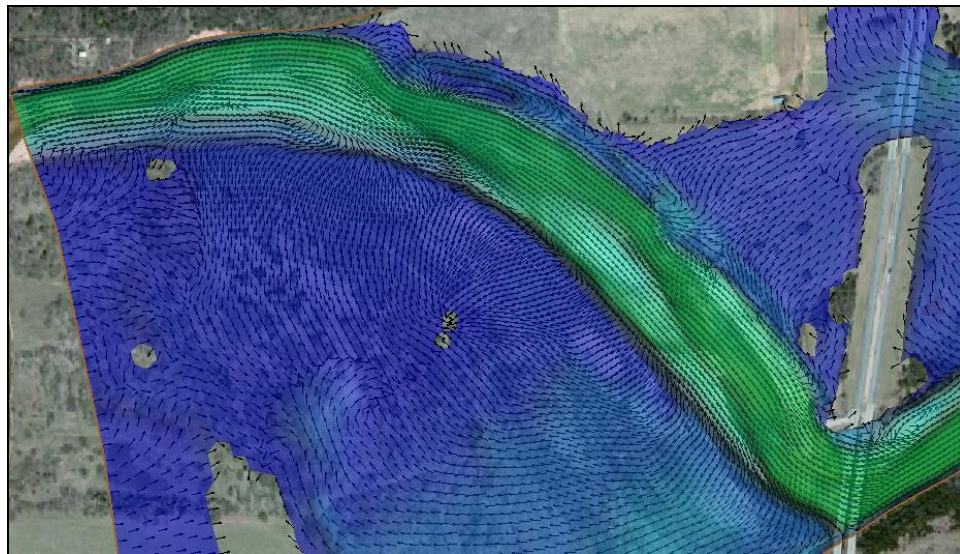The project should appear similar to Figure 3.

Figure 3   Velocity magnitude and velocity vector

To view the vectors on a normalized grid:

1.   Click **Display Options** 🖥 to open the *Display Options* dialog.

2.   Select "2D Mesh" from the list on the left.

3.   On the *Vectors* tab, in the *Vector Display Placement and Filter* section, select "on a grid" from the *Display* drop-down.

4.   Enter "15" for both *X spacing* and *Y spacing*.

5.   Click **OK** to close the *Display Options* dialog.

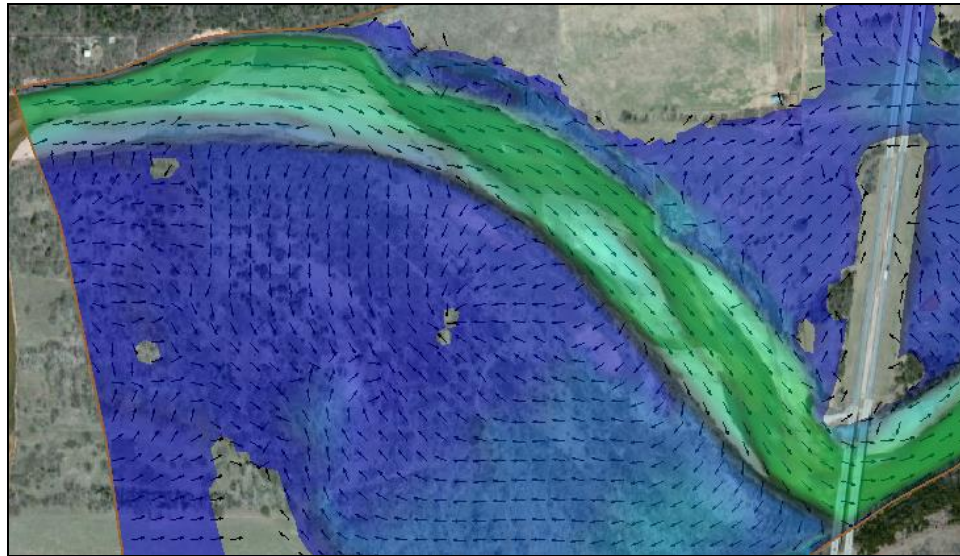The project should appear similar to Figure 4.

Figure 4   Vectors normalized on a grid

This method of displaying vectors is useful when displaying areas with both coarse and refined areas. With the vectors displayed on a grid, the resolution of the vectors displayed either increases or decreases when zooming in or out to maintain the same pixel spacing.

3.4   Film Loop Visualization

In addition to single time steps of contours and vectors, animations can be generated and saved by using the film loop feature.

To create a film loop of the SRH-2D analysis:

1. Select *Data* / **Film Loop…** to bring up the *General Options* page of the *Film Loop Setup* dialog.

2. In the *Select Film Loop Type* section, select *Transient Data Animation Type*.

3. Click **Next** to go to the *Time Options* page of the *Film Loop Setup* dialog.

4. Click **Next** to go to the *Display Options* page of the *Film Loop Setup* dialog.

5. Click **Finish** to close the *Film Loop Setup – Display Options* dialog.

SMS now starts the film loop, adding one frame at a time. Once the last frame has been added to the loop, the Play AVI Application will open and the animation will start playing automatically.

Continue to experiment with the film loop features if desired. Click **Close** at the top right corner of the Play AVI Application window when finished. The film loop has been saved as "sms.avi" in the tutorial directory.

4   Conclusion

This concludes the "SRH-2D Post-Processing" tutorial. Feel free to continue to experiment with these post-processing features in SMS, or exit the program.