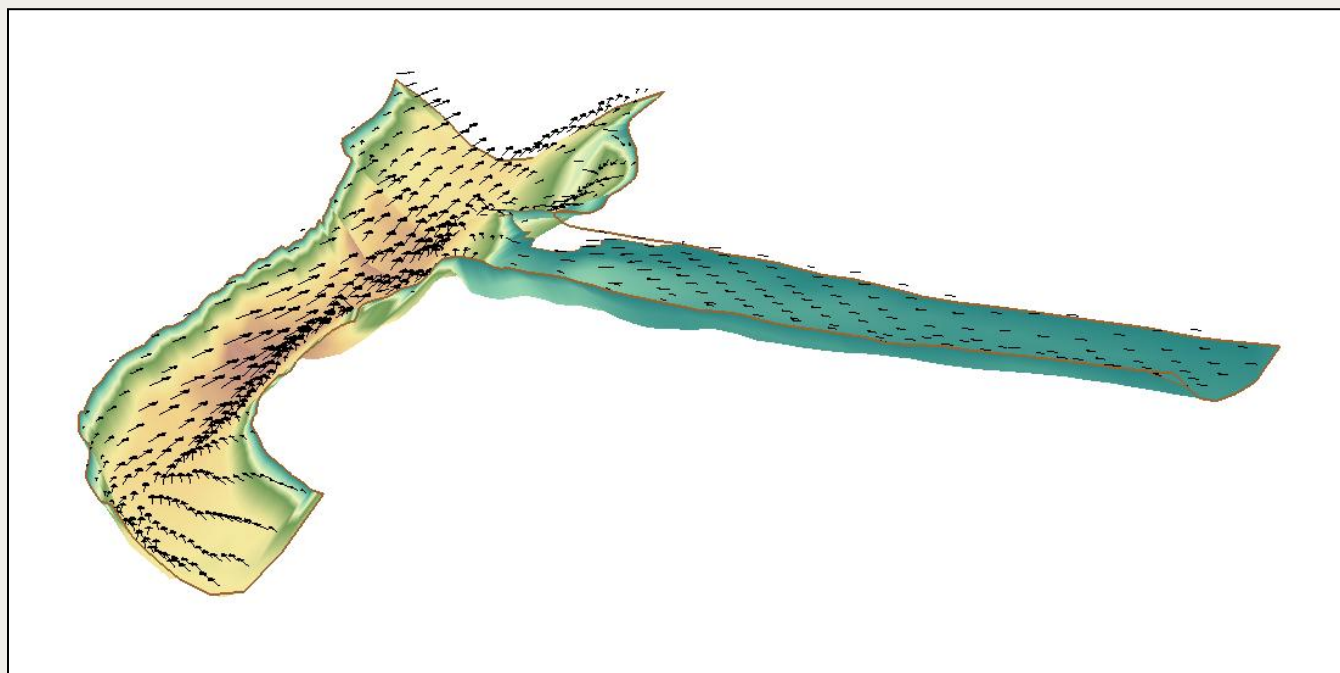




*SMS 14.0 Tutorial*

**Data Visualization**

Using the Display Tools in SMS



**Objectives**

Learn how to import, manipulate, and view solution data.

**Prerequisite Tutorials**

- None

**Required Components**

- SMS Core

**Time**

- 20–30 minutes

|          |                                                             |           |
|----------|-------------------------------------------------------------|-----------|
| <b>1</b> | <b>Introduction .....</b>                                   | <b>2</b>  |
| <b>1</b> | <b>Opening the Geometry and Solution Files .....</b>        | <b>2</b>  |
| <b>2</b> | <b>Creating New Datasets with the Data Calculator .....</b> | <b>3</b>  |
| <b>3</b> | <b>Contours .....</b>                                       | <b>4</b>  |
| 3.1      | Turning on Contours .....                                   | 4         |
| 3.2      | Color Ramp Options .....                                    | 6         |
| <b>4</b> | <b>Vectors .....</b>                                        | <b>7</b>  |
| <b>5</b> | <b>Creating Animations .....</b>                            | <b>9</b>  |
| 5.1      | Creating a Film Loop Animation .....                        | 9         |
| 5.2      | Animating a Functional Surface .....                        | 10        |
| 5.3      | Creating a Flow Trace Animation .....                       | 11        |
| 5.4      | Drogue Plot Animation .....                                 | 12        |
| <b>6</b> | <b>2D Plots .....</b>                                       | <b>14</b> |
| <b>7</b> | <b>Conclusion .....</b>                                     | <b>14</b> |

## 1 Introduction

Visualization increases the applicability and usefulness of the modeling process. Viewing the geospatial data utilized as input and generated as solutions in the process of numerical analysis can be very useful. Extracting data along a line (profile or transect) or at a point from this geospatial data can also be helpful.

A geospatial dataset has one or more numeric values associated with each node in a mesh, cell in a grid, vertex in a scatter set, and so on. Scalar datasets have one value per location. Some examples of scalar datasets include bathymetry, water surface elevation, velocity magnitude, Froude number, energy head, concentration, bed change, wave heights.

Two-dimensional vector datasets have two values for every location (an x component and a y component). Examples of vector datasets include observed wind fields, flow velocities, shear stresses, and wave radiation stress gradients.

Steady state datasets represent a numerical solution where nothing changes with time. Dynamic datasets have data at specific times (time steps) to represent a numerical solution that changes with time.

## 1 Opening the Geometry and Solution Files

SMS opens all supported input and solution files using the *File* | **Open** command.

1. Select *File* | **Open...** to bring up the *Open* dialog.
2. Browse to the *data files\* folder for this tutorial.
3. Select "Project Files (\*.sms)" from the *Files of type* drop-down.
4. Select "data\_visualization.sms" and click **Open** to import the project and exit the *Open* dialog.
5. Select  Velocity\_ft\_p\_s" to make it active.
6. Click **Display Options**  to open the *Display Options* dialog.
7. Select "2D Mesh" from the list on the left.
8. On the *2D Mesh* tab, turn off *Nodes* and *Elements*.
9. Turn on *Contours* and *Vectors*.
10. On the *Contours* tab, in the *Contour method* section, select "Color Fill" from the first drop-down.

11. On the *Vectors* tab, in the *Arrow Options* section, from the *Shaft Length* drop-down, select “Scale length to magnitude”.
12. Enter “4.0” as the *Ratio*.
13. Click **OK** to close the *Display Options* dialog.

The mesh should appear similar to Figure 1.

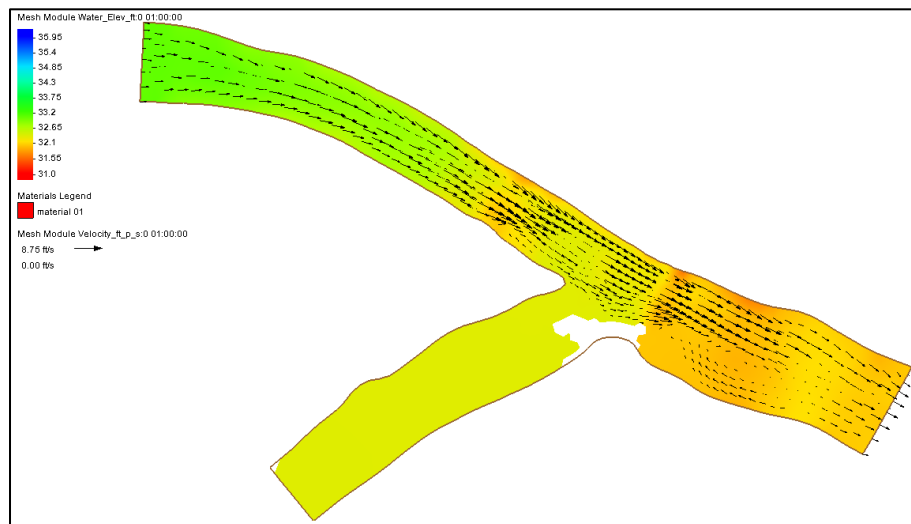


Figure 1 The mesh of the velocity dataset with contours and vectors turned on

## 2 Creating New Datasets with the Data Calculator

SMS uses a tool called the *Data Calculator* for computing new datasets by performing operations on scalar values and existing datasets. In this example, a dataset will be created which contains the Froude number at each node. The Froude number is given by this equation:

$$\text{Froude Number} = \frac{\text{Velocity}}{\sqrt{\text{gravity} * \text{WaterDepth}}}$$

Before creating the Froude dataset, the water depth dataset needs to have negative values removed. Fix the depth dataset by doing the following:

1. Click the **Toolbox**  macro to open the *Toolbox* dialog.
2. In the *Tools* tab, expand the “ Datasets” folder.
3. Double-click on the **Dataset Calculator** tool to open the *Dataset Calculator* dialog.
4. For *Grid source selection*, set to “Mesh Data/data visualization”.
5. For *Data location selection*, set to “Points”.
6. Under the *Dataset table*, click the **Add Row**  button.
7. In row 1, set the *Dataset* to “tribflood (SRH-2D)/Water\_Depth\_ft”.
8. Make certain *Timestep* is set to “All time steps”.
9. For the *Mathematical expression*, enter “where (d1<0, 0.0001, d1)”.
10. For the *Output dataset name* enter “Depth1”.

11. Click **OK** to run the tool.

This creates a new dataset that has had negative depth values removed. Now to create the Froude dataset by doing the following:

1. Click the **Toolbox**  macro to return to the *Toolbox* dialog.
2. In the *Tools* tab, expand the “ Datasets” folder.
3. Double-click on the **Dataset Calculator** tool to open the *Dataset Calculator* dialog.
4. For *Grid source selection*, set to “Mesh Data/data visualization”.
5. For *Data location selection*, set to “Points”.
6. Under the *Dataset table*, click the **Add Row**  button.
7. In row 1, set the *Dataset* to “tribflood (SRH-2D)/Vel\_Mag\_ft\_s”.
8. Make certain *Timestep* is set to “All time steps”.
9. Click the **Add Row**  button.
10. In row 2, set the *Dataset* to “Depth1”.
11. Make certain *Timestep* is set to “All time steps”.
12. For the *Mathematical expression*, enter “ $d1/\sqrt{32.2*d2}$ ”.

The expression is the Froude number equation where “d1” represents the velocity dataset and “d2” represents the water depth dataset.

13. For the *Output dataset name* enter “Froude”.
14. Click **OK** to run the tool.
15. When the tool finishes, click **OK** to close the *Dataset Calculator* window.

The “ Froude” dataset can be contoured and edited just as with any other dynamic scalar dataset. It can also be saved in a generic dataset file. See *SMS Help* for more information on saving datasets.

## 3 Contours

SMS provides several contour options to help visualize datasets.

### 3.1 Turning on Contours

For this tutorial, create contours for the velocity magnitude dataset by doing the following:

1. Select “ Vel\_Mag\_ft\_p\_s” (velocity magnitude) in the Project Explorer.
2. Click on “0 00:05:00” in the *Time steps* list box below the Project Explorer.
3. Click **Display Options**  to bring up the *Display Options* dialog.
4. Select “2D Mesh” from the list on the left.
5. On the *2D Mesh* tab, click **All Off** to turn off all current display options on this tab.
6. Turn on *Contours*, *Mesh boundary*, and *Wet/dry boundary*.

7. On the *Contours* tab, in the *Contour method* section, select “Linear” from the first drop-down.
8. In the *Contour interval* section, select “Number” from the drop-down.
9. Enter “20” in the field to the right.
10. Click **OK** to close the *Display Options* dialog.

The project should appear similar to Figure 2.

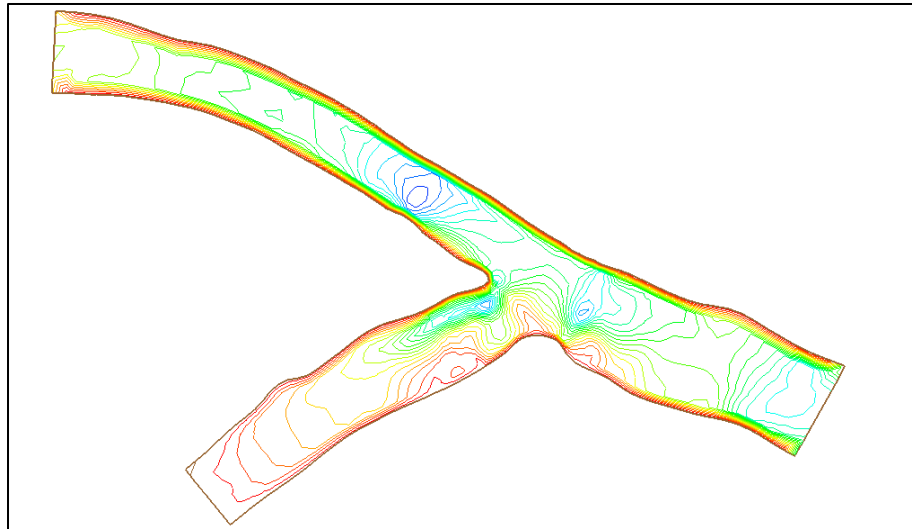


Figure 2 Linear contour display

SMS supports color filled contours as well as color filled with linear contours at the breaks. Do the following to use color filled contours:

11. Right-click on “ Mesh Data” and select **Display Options...** to bring up the *Display Options* dialog.

This is an alternative to using the **Display Options**  macro.

12. Select “2D Mesh” from the list on the left.
13. On the *Contours* tab, in the *Contour method* section, select “Color Fill” from the first drop-down.
14. In the *Contour interval* section, near the bottom, turn on *Fill continuous color range*.

This option causes SMS to blend dataset values rather than use discreet intervals.

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

The project should appear similar to Figure 3.

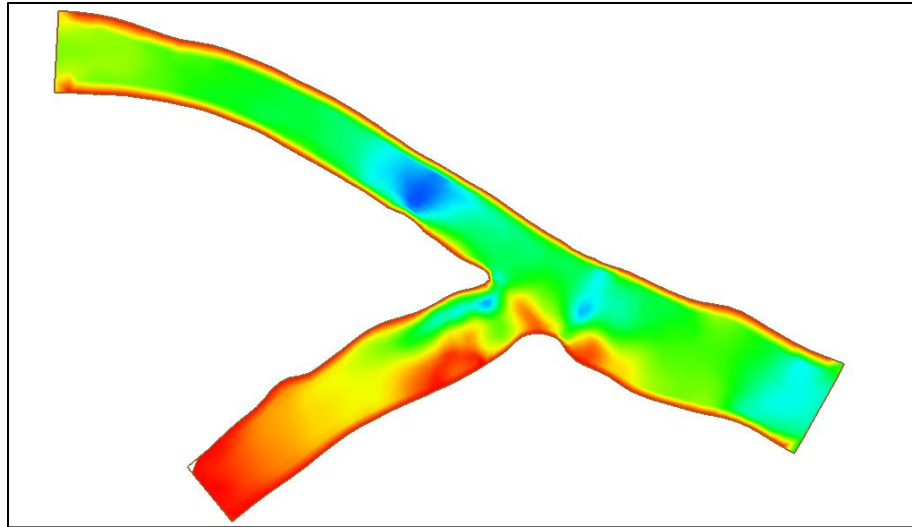


Figure 3 Color filled contour display

### 3.2 Color Ramp Options

The default color ramp in SMS has dark blue for the largest scalar value and dark red for the smallest scalar value. Other color ramps can be useful for visualizing data and can be saved as part of a project or as the default when running SMS.

Do the following to use a different color ramp to better visualize water depth:

1. In the Project Explorer, select “ Water\_Depth\_ft” 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 *Contours* tab, in the *Contour method* section, click **Color Ramp...** to bring up the *Choose color ramp* dialog.
5. Select “Ocean” from the list of color ramps.
6. Click **OK** to exit the *Choose color ramp* dialog.
7. Click **OK** to exit the *Display Options* dialog.

This color ramp shows the deeper areas as dark blue and shallower areas as light blue as seen in Figure 4.

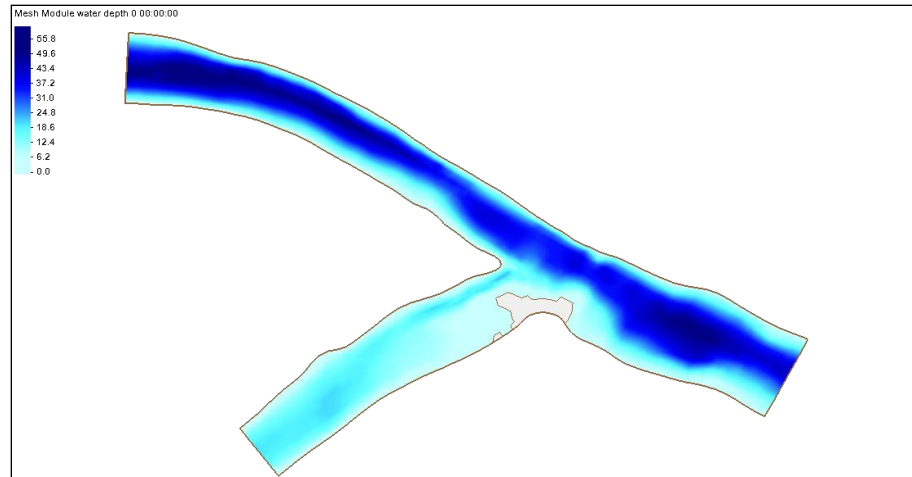


Figure 4 Using the color ramp with the ocean color palette

## 4 Vectors

Vector datasets can be visualized inside of SMS by displaying arrows representing the direction (and optionally the magnitude) of the vector dataset over the mesh.

Do the following to turn on vectors for the velocity magnitude dataset:

1. In the Project Explorer, select “ Vel\_Mag\_ft\_p\_s” 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 on *Vectors*.
5. On the *Contours* tab, in the *Contour method* section, click **Color Ramp...** to bring up the *Choose color ramp* dialog.
6. Select “Hue 1” from the list of color ramps.

This makes red indicate the higher velocities.

7. Click **OK** to exit the *Color Options* dialog.
8. On the *Vectors* tab, in the *Arrow Options* section, from the *Shaft Length* dropdown, select “Define min and max length”.

This scales the length of the arrows based upon the magnitude of the velocity dataset at the arrow location. The minimum dataset magnitude uses the shaft length that is the minimum length. Likewise, the maximum dataset magnitude uses the maximum shaft length.

9. For *Minimum*, enter “10”.
10. For *Maximum*, enter “80”.
11. Click **OK** to close the *Display Options* dialog.

Arrows should now be displayed that show the magnitude and direction of the water currents over the mesh. However, the arrows are so dense that it can be challenging to see details (Figure 5).

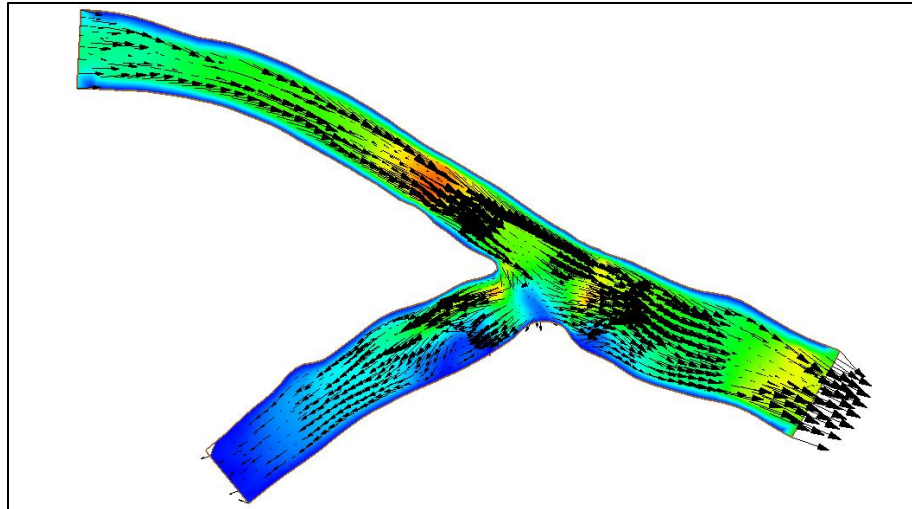


Figure 5 Dense velocity magnitude and direction arrows

To thin out the arrows, follow these steps:

12. Click **Display Options**  to bring up the *Display Options* dialog.
13. Select “2D Mesh” from the list on the left.
14. On the *Vectors* tab, in the *Vector Display Placement and Filter* section, from the *Display* drop-down, select “on a grid”.
15. For both *X spacing* and *Y spacing*, enter “25”.
16. For *Offset*, enter “5.0”.
17. Click **OK** to exit the *Display Options* dialog.

Now the arrows should be evenly distributed over the domain at 25 pixel increments (Figure 6). The z-offset lifts the vectors off the mesh by 5.0 feet. Variations in the shape of the riverbed can hide vectors since they are drawn in three dimensions.

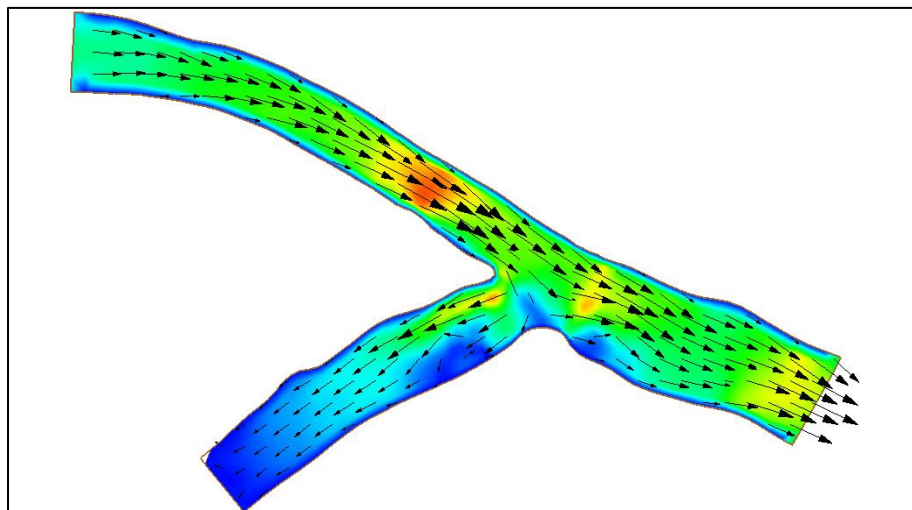


Figure 6 Vector velocity arrows on a grid

Right below where the two branches join, an eddy is formed (indicated by the arrows in the recirculation pattern).



18. **Zoom**  into the eddy area as shown in Figure 7.

After zooming, the vector spacing stays at 25 pixels. Therefore, additional vectors appear illustrating the recirculation pattern.

19. When done reviewing the vectors, **Frame**  the project.

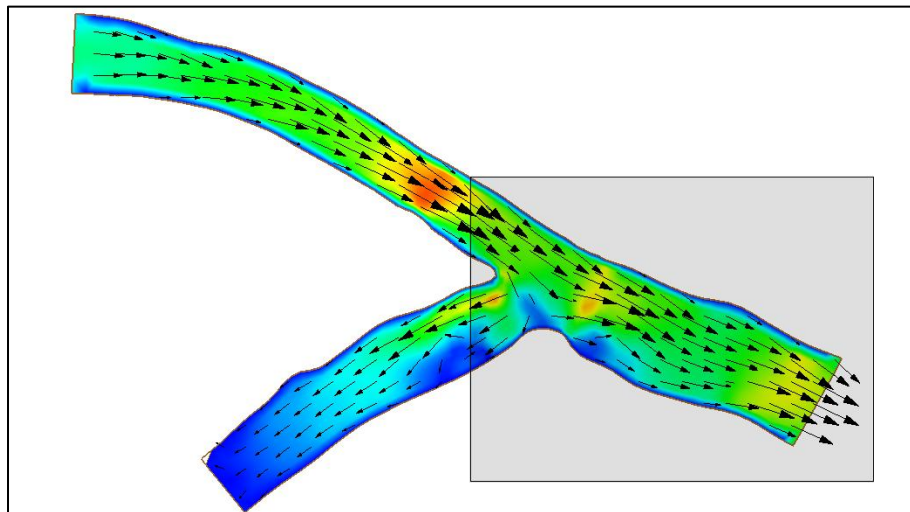


Figure 7 Zoom into this area

## 5 Creating Animations

A film loop is an animation created by SMS to display changes in datasets through time. Flow trace and particle trace animations are a special type of film loop which use vector datasets to trace the path that particles of water will follow through the flow system. Only the visible portion of the mesh will be included in the film loop when it is created.

### 5.1 Creating a Film Loop Animation

The following film loop will show how the velocity changes through time. To create and run the film loop, do the following:

1. In the Project Explorer, while pressing the *Shift* key, select both the scalar ( "Vel\_Mag\_ft\_p\_s") and the vector ( "Velocity\_ft\_p\_s") datasets.
2. Select **Data | Film Loop...** to bring up the *General Options* page of the *Film Loop Setup* dialog.
3. In the *Select Film Loop Type* section, select *Transient Data Animation*.
4. In the *Film Loop Files* section, turn on *Create Animation File*.
5. Click **File Browser**  to bring up the *Save* dialog.
6. Enter "velocity.mp4" as the *File name*.
7. Click **Save** to close the *Save* dialog.
8. Click **Next** to go to the *Time Options* page of the *Film Loop Setup* dialog.

For this tutorial, the default settings match the solution time step and duration needed.

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

The last page of the dialog allows the display options to be modified and a clock to be specified.

10. In the *Options* section, click **Clock Options...** to bring up the *Clock Options* dialog.
11. In the *Location* section, select “Top Right Corner” from the drop-down.
12. Click **OK** to close the *Clock Options* dialog.
13. Click **Finish** to close the *Film Loop Setup* dialog and generate the film loop.

SMS displays each frame of the film loop as it is being created. The generated film loop illustrating a storm hydrograph coming down the tributary will be saved in the MP4 file format.

## 5.2 Animating a Functional Surface

Functional surfaces can be used to visualize datasets as a surface with the elevation at each node being the value of the dataset plus a constant offset. A functional surface can be used to display the water surface.

Do the following to turn on the functional surface:

1. Click **Display Options**  to bring up the *Display Options* dialog.
2. Select “2D Mesh” from the list on the left.
3. On the *2D Mesh* tab, turn off *Vectors* and turn on *Functional Surface*.
4. Click **Options...** to open the *Functional Surface Options* dialog.
5. In the *Data Set* section, select *User defined dataset* to bring up the *Select Dataset* dialog.
6. In the *Select* section, select the “Water\_Elev\_ft” dataset and click **Select** to close the *Select Dataset* dialog.
7. Click **OK** to close the *Functional Surface Options* dialog.
8. Click **OK** to close the *Display Options* dialog.
9. Select *Display | View | Oblique* to change the view to an oblique (3D) view.

The oblique view gives a better angle to the animation. For consistency, the angle can be set to a specific view angle by doing the following:

10. Click **Display Options**  to bring up the *Display Options* dialog.
11. Select “General” from the list on the left.
12. On the *View* tab, in the *View angle* section, for *Bearing*, enter “43.0” and for *Dip*, enter “22.0”.
13. Under *Looking at point*, for X, Y, and Z, respectively, enter “17275.0”, “13900.0”, and “5.75”.
14. Under *Defined view bounds size*, select *Width* and enter “3200.0”.
15. Then click **OK** to close the *Display Options* dialog.

The functional surface of the water surface should appear over the bathymetry, shaded with the velocity magnitude contours as shown in Figure 8. The selected values are to illustrate the oblique view. The **Rotate**  tool can be used to change the view as well.

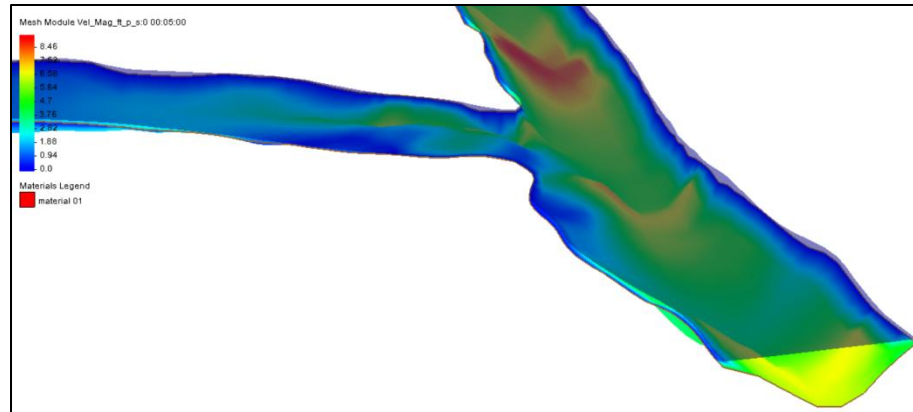


Figure 8 Functional surface with velocity magnitude contours

This surface can be animated to show the change in water surface elevation through time. In the case of this water surface, there are not large changes. However, the water level does rise in the tributary and some flooding does occur.

16. Repeat steps 1–13 in section 5.1 to animate the functional surface, using “wse.mp4” as the *File name* in step 6.
17. **Frame**  the project.

### 5.3 Creating a Flow Trace Animation

A flow trace animation can be created if a vector dataset has been opened. The flow trace simulates spraying the domain with colored dye droplets and watching the color flow through the domain. Steady state vector fields can be used in a flow trace animation to show flow direction trends. For dynamic vector fields, the flow trace animation can trace a single time-step or it can trace the changing flow field.

A flow trace generally takes longer to generate than scalar or vector animations. As the window gets bigger and shows more of the model, the animation gets larger and requires more memory to generate it. If problems are encountered with this operation, decrease the size of the SMS window and try again.

To create and run a flow trace film loop, do the following:

1. Switch back to **Plan View** .
2. **Zoom**  in on the area of the junction of the two reaches.
3. Select **Data | Film Loop...** to bring up the *General Options* page of the *Film Loop Setup* dialog.
4. In the *Film Loop Files* section, click the **File Browser**  button to open the *Save* dialog.
5. Enter “flowtrace.mp4” as the *File name* and click **Save** to close the *Save* dialog.
6. In the *Select Film Loop Type* section, select *Flow Trace*.
7. Click **Next** to go to the *Time Options* page of the *Film Loop Setup* dialog.
8. Click **Next** to accept the default settings and go to the *Flow Trace Options* page.
9. In the *Options* section, for *Particles per object* enter “0.5”, and for *Decay ratio*, enter “0.1”.
10. Click **Next** to go to the *Display Options* page of the *Film Loop Setup* dialog.

11. Click **Finish** to generate the animation.

After a few moments, the first frame of the flow trace animation will appear on the screen. As before, the frames are generated one at a time, and the Graphics Window will show which frame is being created. When the flow trace has been created, the MP4 file can be viewed in a media application.

## 5.4 Drogue Plot Animation

---

Droque plot animations are similar to flow trace animations, except that they specify where particles will start. A particle-droque coverage defines the starting location for each particle.

To create this coverage:

1. Select the  "Water\_Elev\_ft" dataset to make it active.
2. **Zoom**  into the area shown in Figure 9.
3. Click **Display Options**  to open the *Display Options* dialog.
4. Select "2D Mesh" from the list on the left.
5. On the *2D Mesh* tab, turn off *Functional surface* and click **OK** to close the *Display Options* dialog.
6. In the Project Explorer, select the "Area Property" coverage to switch to the **Map**  module.
7. Right-click on "Area Property" and select *Type | Generic | Particle-Droque*.
8. Using the **Create Feature Arcs**  tool, create a feature arc across the top branch of the river, double-clicking on the other side to complete the arc (see Figure 9).
9. Repeat step 8 to create another arc across the lower left branch of the river (see Figure 9).
10. Using the **Select Feature Arcs**  tool, press the *Shift* key while selecting each arcs.
11. Select *Feature Objects | Redistribute Vertices...* to bring up the *Redistribute Vertices* dialog.
12. In the *Arc Redistribution* section, for *Specify* select "Number of segments" from the drop-down.
13. For *Number of segments*, enter "20".
14. Click **OK** to close the *Redistribute Vertices* dialog.
15. Using the **Create Points**  tool, create three points in the downstream branch of the river as shown in Figure 9.

The arcs and points should appear similar to Figure 9. For the drogue plot animation, one particle will be created at each feature point and each feature arc vertex.

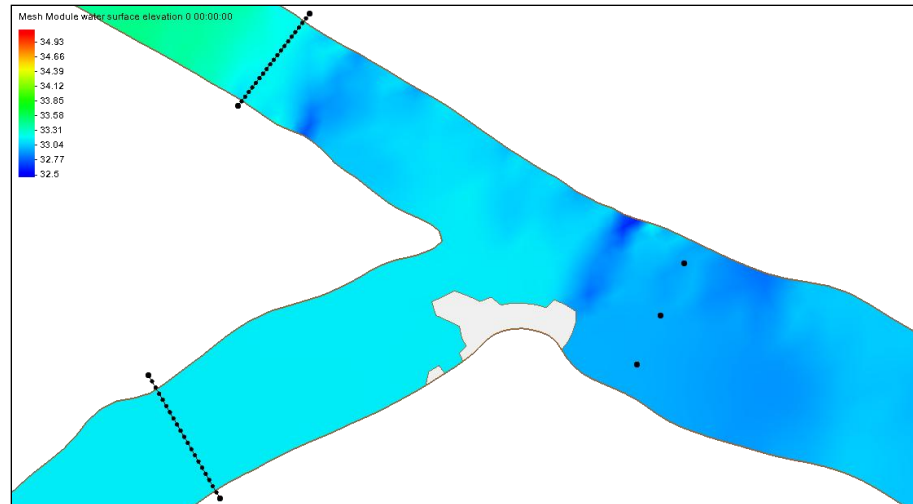


Figure 9 Arcs and points for drogue plot animation

To create the drogue plot animation:

16. Switch to the **Mesh**  module.
17. Select **Data | Film Loop...** to bring up the *General Options* page of the *Film Loop Setup* dialog.
18. In the *Film Loop Files* section, click the **File Browser**  button to bring up the *Save* dialog.
19. Enter “drogue.mp4” as the *File name* and click **Save** to close the *Save* dialog.
20. In the *Select Film Loop Type* section, select *Drogue Plot*.
21. Click **Next** to go to the *Time Options* page of the *Film Loop Setup* dialog.
22. In the *Filmloop Time* section, in the *Run Simulation* field enter “5.0” and select “hours” from the drop-down to the right.
23. For the *Specify Time Step Size* field, enter “5.0” and select “minutes” from the drop-down menu to the right.
24. Click **Next** to go to the *Drogue Plot Options* page of the *Film Loop Setup* dialog.
25. In the *Color Options* section, select *Distance traveled*.
26. For *Maximum value*, enter “1500.0”.
27. In the *Statistical Report* section, turn on *Write report*.
28. Click **Next** to go to the *Display Options* page of the *Film Loop Setup* dialog.
29. Click **Finish** button to close the *Film Loop Setup* dialog and generate the animation file.

Frame 3 of the animation should appear similar to Figure 10.

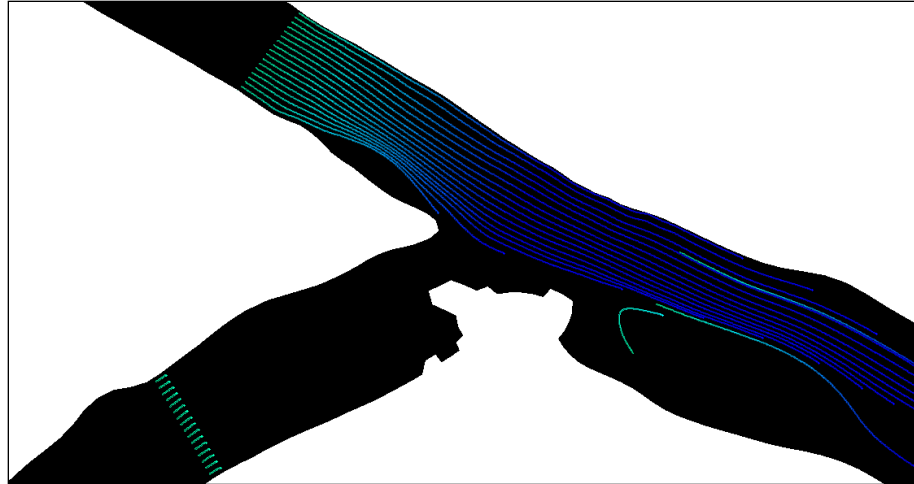


Figure 10 Frame 3 of the drogue plot animation

The drogue plot animation generates a report by turning on the option in step number 3 above. This file is saved in the same directory as the animation. To see this report, do the following:

30. Choose *File* | **View Data File...** to bring up the *Open* dialog.
31. Select “drogues.pdr” and click **Open** to exit the *Open* dialog and bring up the *View Data File* dialog. If the *Never ask this again* option was previously turned on, this dialog will not appear. In that case, skip to step 33.
32. Select the desired text editor from the *Open with* drop-down and click **OK** to close the *View Data File* dialog and open “drogues.pdr” in the selected text editor.
33. Review the contents of “drogues.pdr,” and exit the text editor once finished.

## 6 2D Plots

---

Plots can be created to help visualize the data. Plots are created using the observation coverage in the map module. See the tutorial “SMS Observation” to learn how to use the observation coverage.

## 7 Conclusion

---

This concludes the “Data Visualization” tutorial. Feel free to continue experimenting with the SMS interface or exit the program.