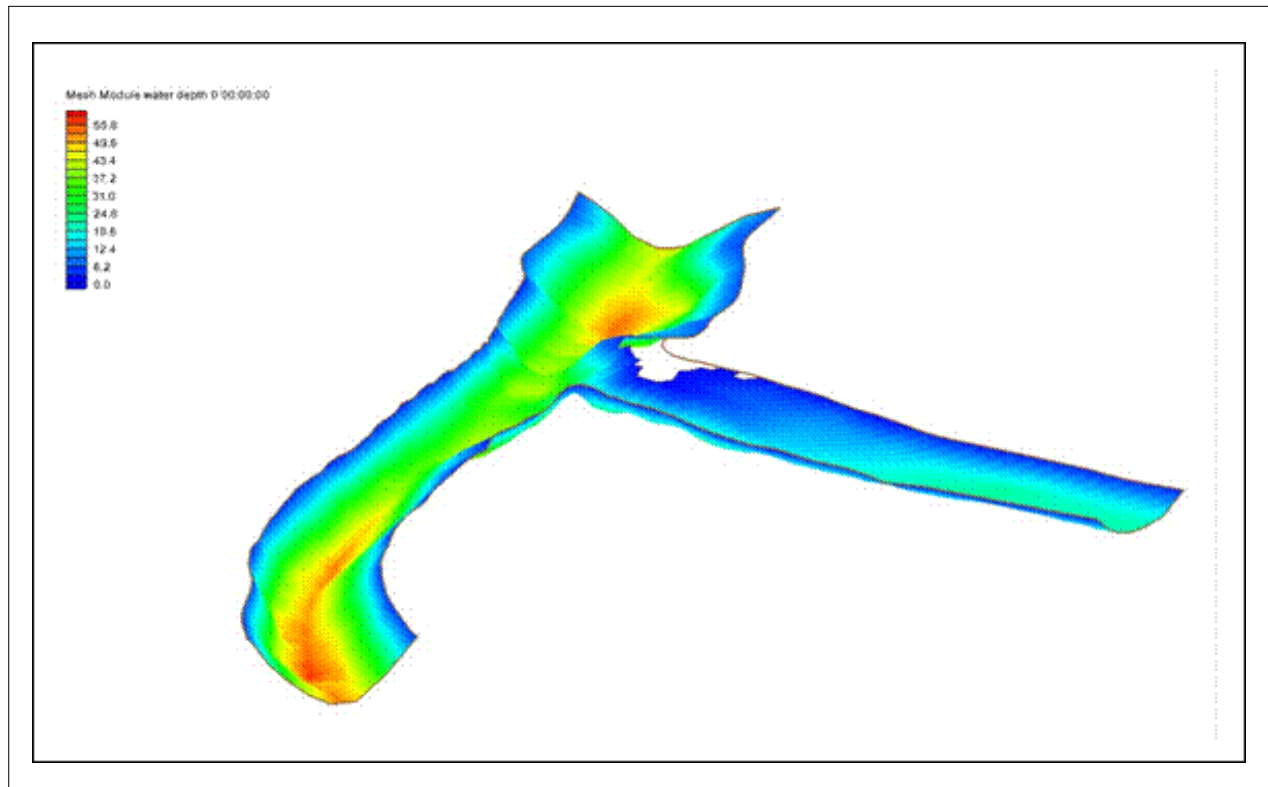


## SMS 13.0 Tutorial

### Data Visualization



### Objectives

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

### Prerequisites

- None

### Requirements

- GIS Module
- Map Module

### Time

- 20–30 minutes

**AQUAVEO™**

<b>1</b>	<b>Introduction.....</b>	<b>2</b>
<b>2</b>	<b>Opening the Geometry and Solution Files .....</b>	<b>2</b>
<b>3</b>	<b>Creating New Datasets with the Data Calculator .....</b>	<b>3</b>
<b>4</b>	<b>Contours .....</b>	<b>4</b>
4.1	Turning on Contours .....	4
4.2	Color Ramp Options .....	6
<b>5</b>	<b>Vectors .....</b>	<b>7</b>
<b>6</b>	<b>Creating Animations.....</b>	<b>9</b>
6.1	Creating a Film Loop Animation .....	9
6.2	Animating a Functional Surface .....	11
6.3	Creating a Flow Trace Animation.....	12
6.4	Droque Plot Animation .....	14
<b>7</b>	<b>2D Plots .....</b>	<b>16</b>
<b>8</b>	<b>Conclusion .....</b>	<b>16</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.

## 2 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” 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* and turn on *Contours* and *Vectors*.
9. On the *Contours* tab, in the *Contour method* section, select “Color Fill” from the first drop-down.
10. On the *Vectors* tab, in the *Arrow Options* section, select “Scale length to magnitude” from the *Shaft Length* drop-down
11. Enter “4.0” as the *Ratio*.
12. Click **OK** to close the *Display Options* dialog.

The mesh should appear similar to Figure 1.

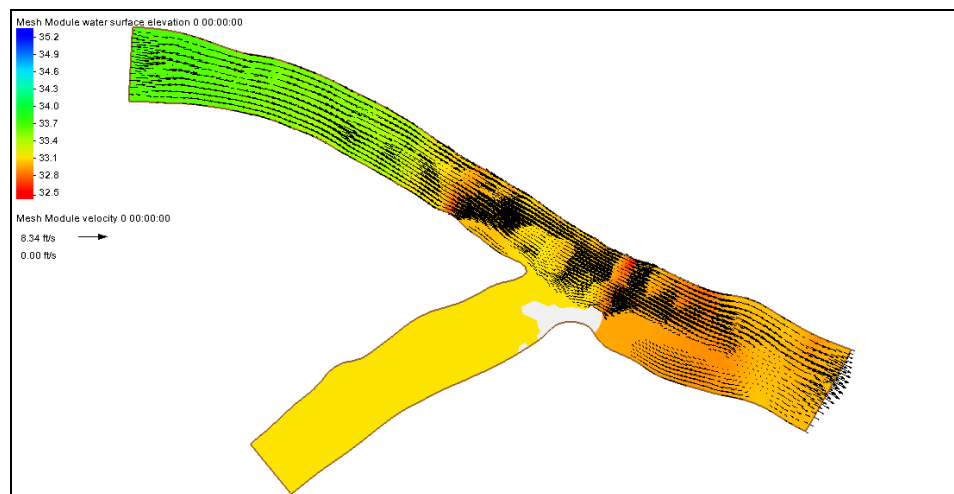


Figure 1 The mesh of the velocity dataset with contours and vectors turned on

### 3 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:

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

Do the following to create the Froude number dataset:

1. Select *Data / Dataset Toolbox...* to bring up the *Dataset Toolbox* dialog.
2. In the *Tools* section, select “Data Calculator” from the tree.

3. In the *Data Calculator* section, in the *Datasets* section, select the “d2. velocity mag” dataset.
4. In the *Time Steps* section, turn on *Use all time steps*.
5. Click **Add to Expression** button at the bottom of the *Datasets* section.

In the *Calculator* section, “d2:all” will appear in the calculation field. The “d2” corresponds to the “velocity mag” dataset, and “all” signifies all time steps.

6. Click the **divide** “/” button.
7. Click the **sqrt** button.

The highlighted “??” text is just a placeholder to indicate that something should be placed there.

8. Enter “32.2” to replace “??” for the constant g.
9. Click the **multiply** “\*” button.
10. In the *Datasets* section, select the “d3. water depth” dataset and click **Add to Expression**.

The expression in the *Calculator* section should now read: “d2:all/sqrt(32.2\*d3:all)”, where “d2” represents the velocity dataset and “d3” represents the water depth dataset. This expression can also be typed in directly.

11. Enter “Froude” as the *Output dataset name*.
12. Click **Compute**.

SMS takes a few moments to perform the computations, and then the new “Froude” dataset will appear in the *Datasets* window.

13. Click **Done** to exit the *Dataset Toolbox* dialog.

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.

## 4 Contours



---

SMS provides several contour options to help visualize datasets.

### 4.1 Turning on Contours

---

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

1. Select “ velocity mag” (velocity magnitude) in the Project Explorer.
2. Click on “0 00:00:00” 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 and enter “20” in the field to the right.
9. Click **OK** to close the *Display Options* dialog.

The project should appear similar to Figure 2.

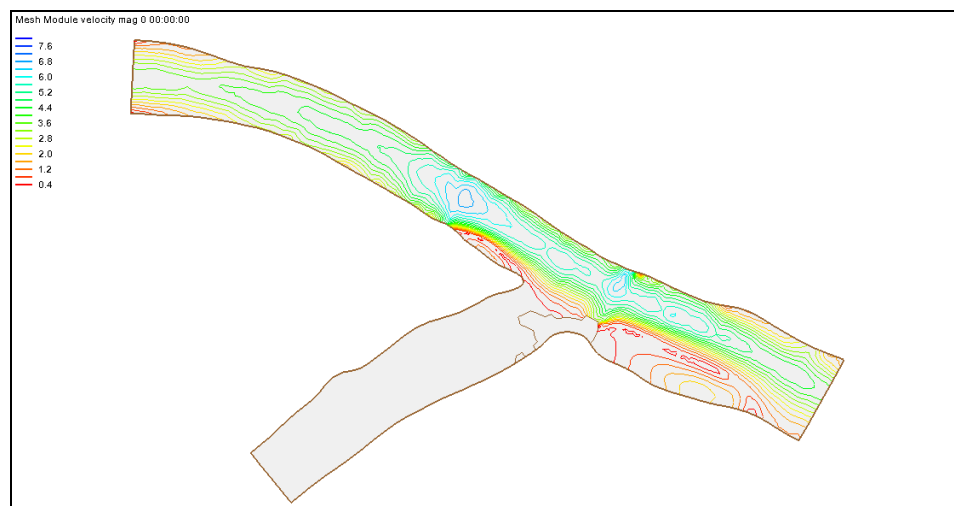




Figure 2 Linear contour display

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

1. 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.

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

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

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

The project should appear similar to Figure 3.

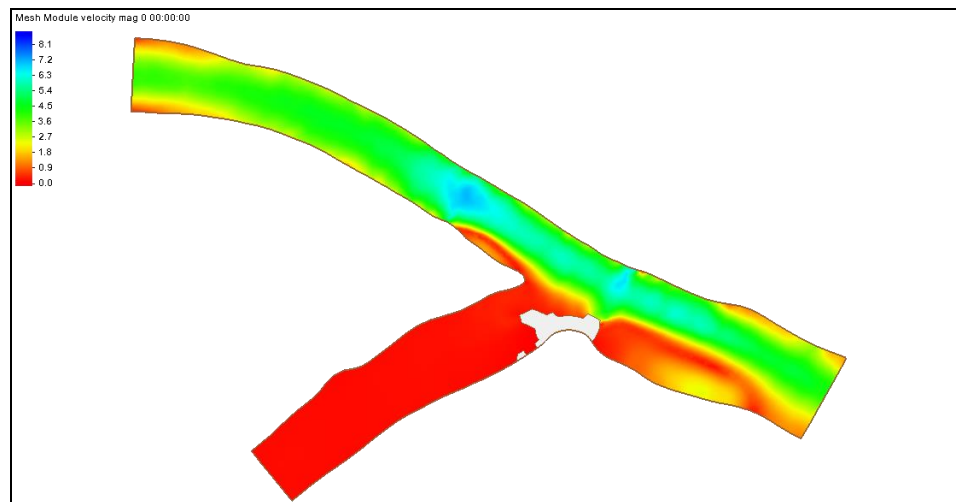




Figure 3 Color filled contour display

## 4.2 Color Ramp Options

The default color ramp in SMS has dark blue for the largest scalar value and a 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 depths:

1. Select “ water depth” 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 *Contours* tab, in the *Contour method* section, click **Color Ramp...** to bring up the *Color Options* dialog.
5. In the *Palette Method* section, select *User defined*.
6. In the *User Defined Palettes* section, click **New Palette** to bring up the *New Palette* dialog.
7. In the *Initial Color Ramp* section, select “Ocean” from the top drop-down.
8. Click **OK** to exit the *New Palette* dialog.
9. Click **OK** to exit the *Color Options* dialog.
10. 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.

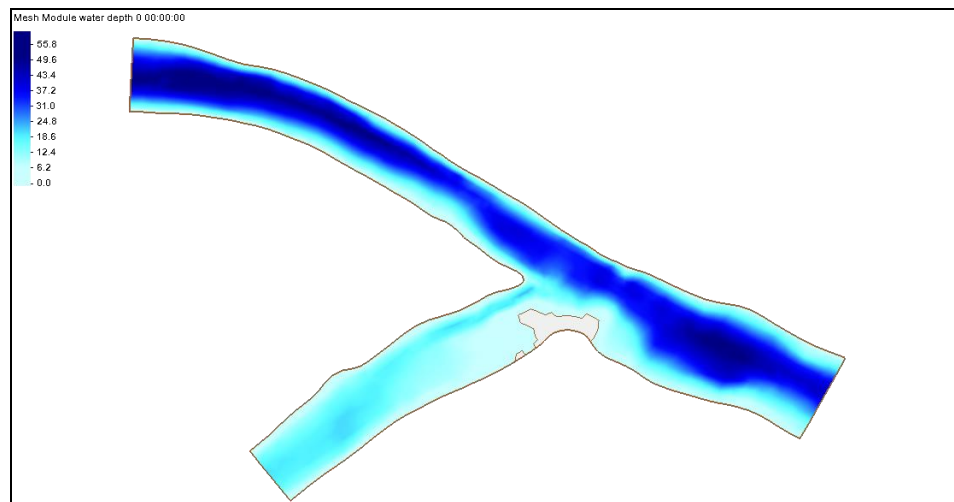




Figure 4 Using the color ramp with the ocean color palette

## 5 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. Select “ velocity mag” 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 on *Vectors*.
5. On the *Contours* tab, in the *Contour method* section, click **Color Ramp...** to bring up the *Color Options* dialog.
6. In the *Palette Method* section, select *Hue ramp*.
7. In the *Palette Preview* section, click **Reverse**.

This makes red indicate the higher velocities.

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

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.

10. Enter “10” as the *Minimum*.
11. Enter “80” as the *Maximum*.

12. 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 is a mess (Figure 5).

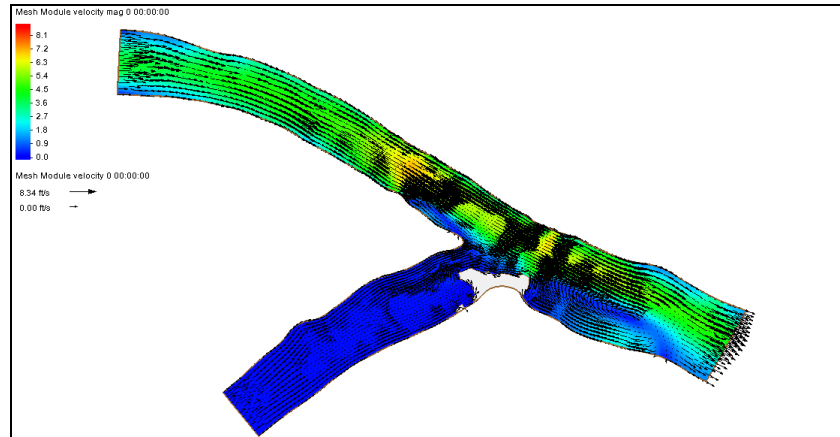



Figure 5 Dense velocity magnitude and direction arrows

To thin out the arrows, follow these steps:

1. Click **Display Options**  to bring up 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 “25” for both *X spacing* and *Y spacing*.
5. Enter “5.0” as the *Offset*.
6. 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 river bed can hide vectors since they are drawn in three dimensions.

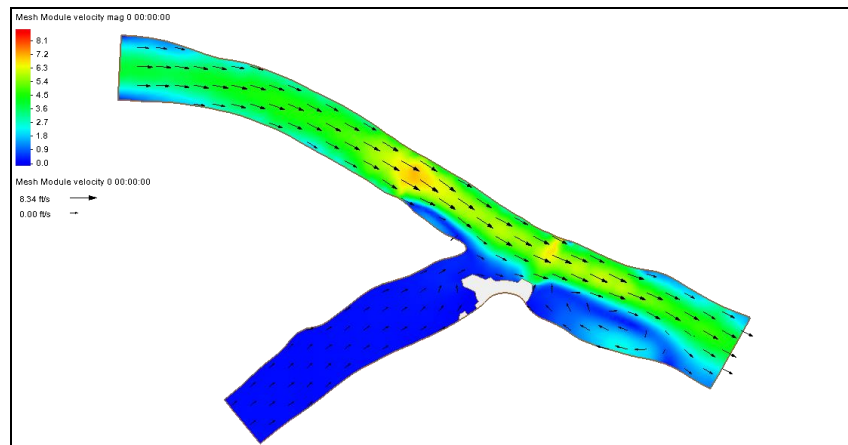




Figure 6 Vector velocity arrows on a grid



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

7. **Zoom**  in to 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.

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

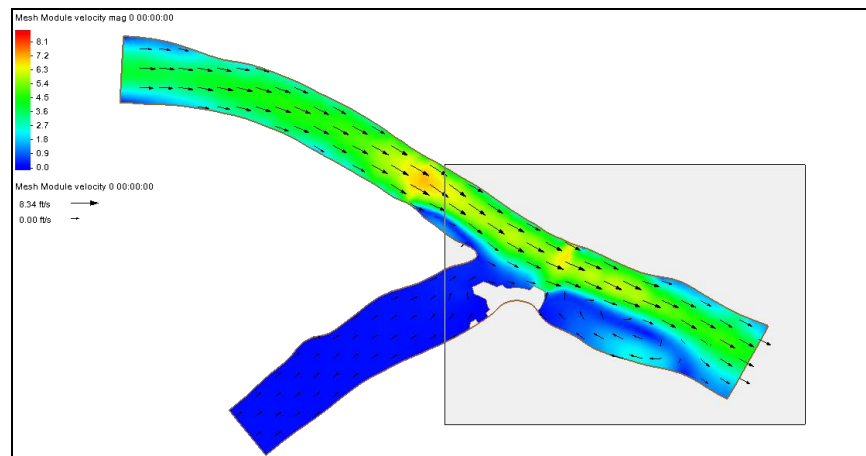





Figure 7 Zoom into this area

## 6 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.

### 6.1 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. While pressing the *Shift* key, select both the scalar (" velocity mag") and the vector (" velocity") datasets in the Project Explorer.
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 AVI File*.
5. Click **File Browser**  to bring up the *Save* dialog.
6. Enter "velocity.avi" 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. When the film loop has been fully generated, the external Play AVI Application (PAVIA) will launch. This application contains the following controls:

- **Play**  – Starts the playback animation. During the animation, the speed and play mode can be changed.
- **Speed** slider bar – Increases or decreases the playback speed. The speed depends on the computer being used.
- **Frame** slider bar – Used to jump to a specific frame of an animation.
- **Stop**  – Stops the playback of the animation.
- **Step**  – Allows manually stepping to the next frame. It only works when the animation is stopped.
- **Loop**  – Automatically restarts the animation when the end of the film loop has been reached.
- **Back/forth**  – Shows the film frames in reverse order when the end of the film loop has been reached.

The generated film loop illustrating a storm hydrograph coming down the tributary will be saved in the AVI file format. AVI files can be used in software presentation packages, such as Microsoft PowerPoint or WordPerfect Presentations. A saved film loop may be opened from inside SMS or directly from inside PAVIA. “Pavia.exe” is located in the SMS installation directory and can be freely distributed.


14. When done viewing the animation, click the  in the top right corner to close the PAVIA window and return to SMS.

## 6.2 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 as follows 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 *Dataset* section, select *User defined dataset* to bring up the *Select Dataset* dialog.
6. In the *Select* section, select the “water surface elevation” 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 views gives a better angle to for 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, enter “43.0” as the *Bearing* and “22.0” as the *Dip*.
13. Under *Looking at point*, enter “17275.0”, “13900.0”, and “5.75” for *X*, *Y*, and *Z*, respectively.
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.

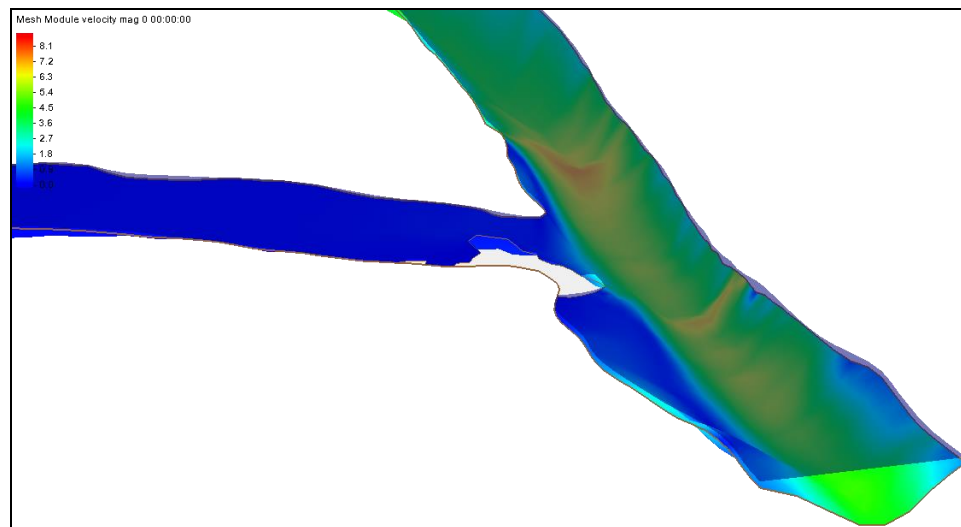




Figure 8 Functional surface with velocity magnitude contours

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 huge changes. However, the water level does rise in the tributary and some flooding does occur.



16. To view the animation, repeat steps 1–13 in section 6.1 to animate the functional surface, using “wse.avi” as the *File name* in step 6.
17. When done viewing the animation, click the  in the top right corner to close the PAVIA window and return to SMS.
18. **Frame**  the project.


### 6.3 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 bring up the *Save* dialog.
5. Enter “flowtrace.avi” 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 time settings and go to the *Flow Trace Options* page of the *Film Loop Setup* dialog.
9. In the *Options* section, enter “0.5” as the *Particles per object*.
10. Enter “0.1” as the *Decay ratio*.
11. Leave the other options at the defaults and click **Next** to go to the *Display Options* page of the *Film Loop Setup* dialog.
12. Click **Finish** to generate the animation and open the external Play AVI Application window.

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, it is launched in a new window, just as with the previous animation (Figure 9). The flow trace can be viewed using the same controls available for the other film loop animation.

13. When done viewing the flow trace animation, click the  in the top right corner to close the PAVIA window and return to SMS.

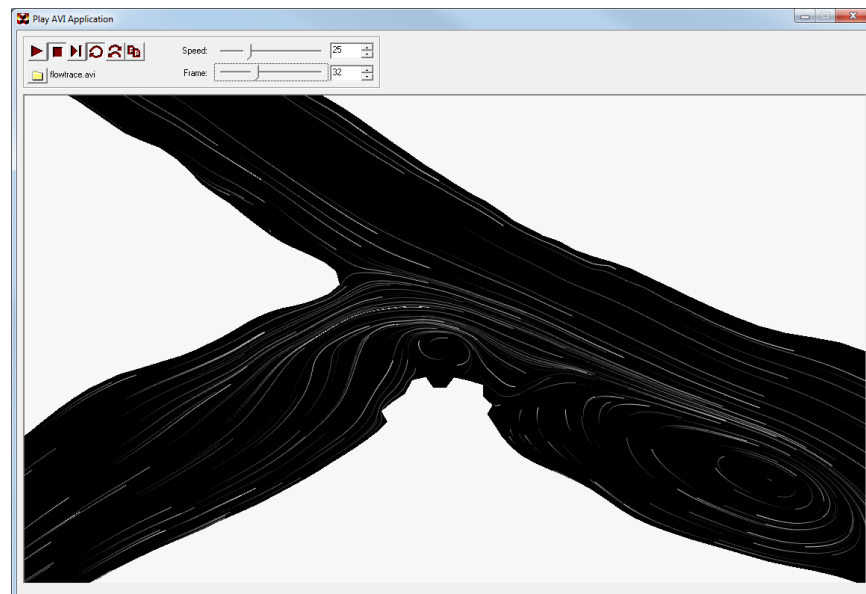











Figure 9 One frame from the *ld1* flow trace

## 6.4 Drogue Plot Animation

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

To create this coverage:

1. Select the “ water surface elevation” dataset to make it active.
2. **Zoom**  into the area shown in Figure 10.
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. Select the “ Area Property” coverage in the Project Explorer to switch to the **Map**  module.
7. Right-click on “ Area Property” and select *Type | Generic | Particle-Drogue*.
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 10).
9. Repeat step 8 to create another arc across the lower left branch of the river (see Figure 10).
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, select “Number of segments” from the *Specify* drop-down.
13. Enter “20” as the *Number of segments*.
14. Click **OK** to close the *Redistribute Vertices* dialog.
15. Using the **Create Points**  tool, create three individual points in the downstream branch of the river as shown in Figure 10.

The arcs and points should appear similar to Figure 10. For the drogue plot animation, one particle will be created at each feature point and each feature arc vertex.

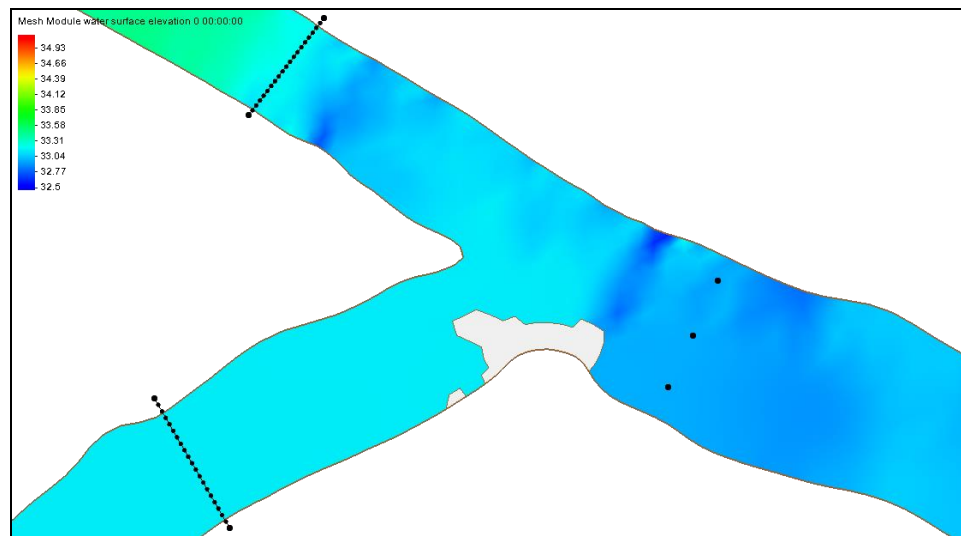




Figure 10 Arcs and points for drogue plot animation

To create the drogue plot animation:

1. Switch to the **Mesh**  module.
2. Select *Data / Film Loop...* to bring up the *General Options* page of the *Film Loop Setup* dialog.
3. In the *Film Loop Files* section, click the **File Browser**  button to bring up the *Save* dialog.
4. Enter “drogue.avi” as the *File name* and click **Save** to close the *Save* dialog.
5. In the *Select Film Loop Type* section, select *Drogue Plot*.
6. Click **Next** to go to the *Time Options* page of the *Film Loop Setup* dialog.
7. In the *Filmloop Time* section, enter “5.0” in the *Run Simulation For* field and select “hours” from the drop-down to the right of that field.
8. Select *Specify Time Step Size*, enter “5.0” in the field, and select “minutes” from the drop-down to the right of that field.
9. Click **Next** to go to the *Drogue Plot Options* page of the *Film Loop Setup* dialog.
10. In the *Color Options* section, select *Distance traveled* and enter “1500.0” in the *Maximum value* field to the right.
11. In the *Statistical Report* section, turn on *Write report*.
12. Click **Next** to go to the *Display Options* page of the *Film Loop Setup* dialog.
13. Click **Finish** button to close the *Film Loop Setup* dialog, generate the animation, and open the external Play AVI Application window.

Frame 3 of the animation should appear similar to Figure 11.

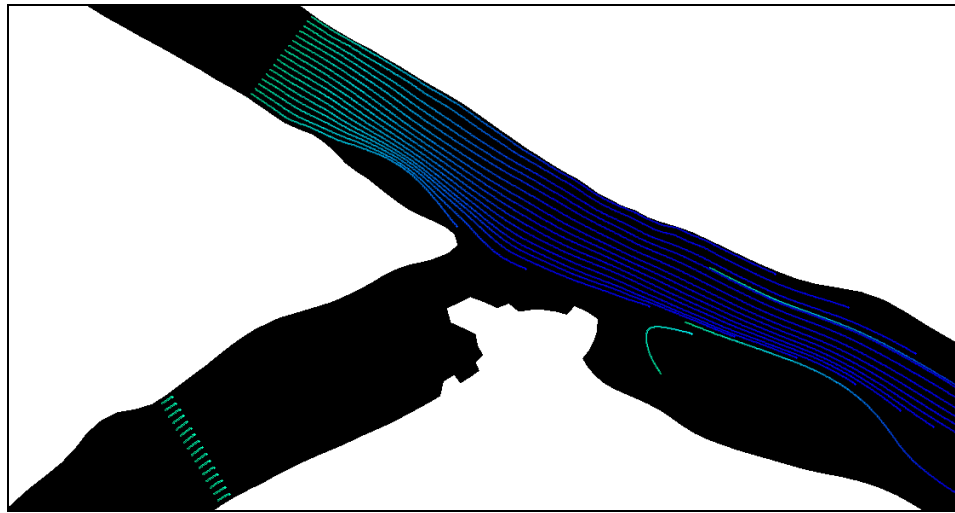



Figure 11 Frame 3 of the drogue plot animation

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:

1. When done viewing the drogue plot animation, click the  in the top right corner to close the PAVIA window and return to SMS.
2. Choose **File | View Data File...** to bring up the *Open* dialog.
3. 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 5.
4. 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.
5. Review the contents of “drogues.pdr”, and exit the text editor once finished.

## 7 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.

## 8 Conclusion

---

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