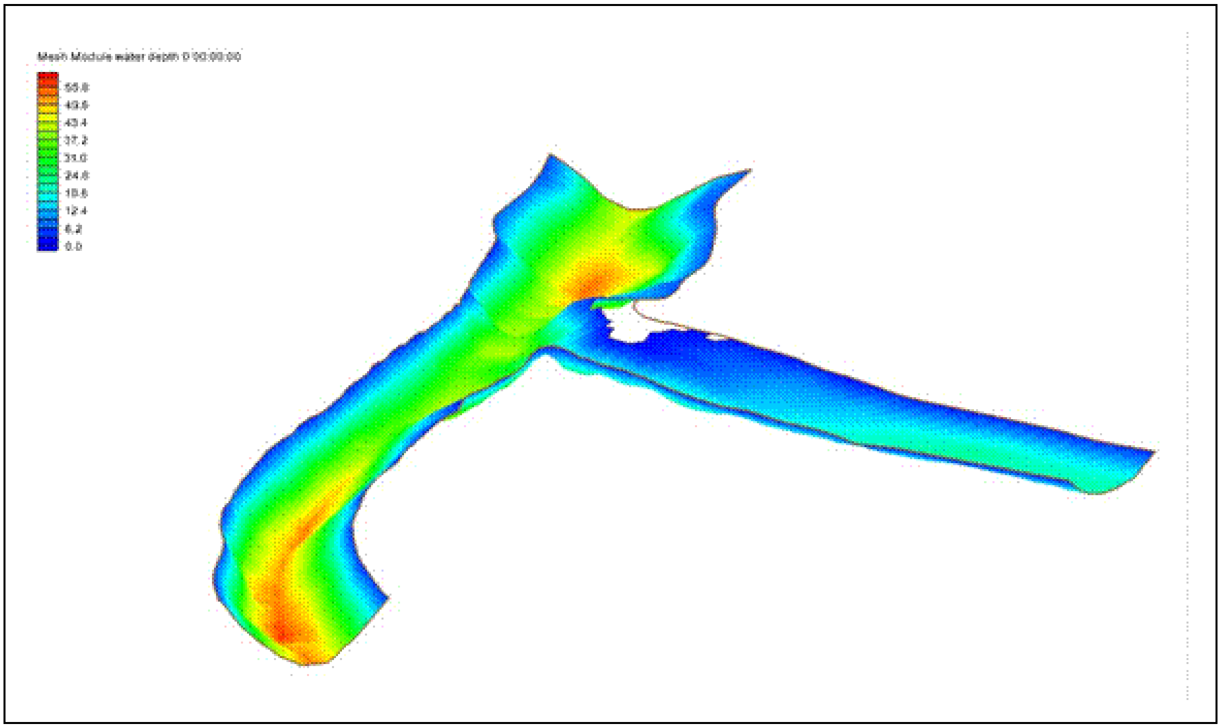
v. 12.1

SMS 12.1 Tutorial

Data Visualization



Objectives

It is useful to view the geospatial data utilized as input and generated as solutions in the process of numerical analysis. It is also helpful to extract data along a line (profile or transect) or at a point from this geospatial data. This *visualization* increases the applicability and usefulness of the modeling process. This lesson will go over how to import, manipulate, and view solution data.

Prerequisites

- None

Requirements

- Map Module
- Mesh Module

Time

- 30–60 minutes

1 Datasets

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, etc. Scalar datasets have one value per location. Two-dimensional vector datasets have two values for every location (an x component and a y component). Examples of scalar datasets include bathymetry, water surface elevation, velocity magnitude, Froude number, energy head, concentration, bed change, wave heights and many more. 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. Select the file “data_visualization.sms” in the *data files* folder for this tutorial.
3. Click **Open** to import the mesh data and solution datasets.

SMS displays the datasets as contours and vectors. To be consistent, do the following:

1. Select the “velocity” dataset to make it active.
2. Use the **Display Options** macro to open the *Display Options* dialog.
3. Select *2D Mesh* from the list on the left then uncheck *Nodes* and check the *Contours* and *Vectors* options.
4. Under the *Contours* tab, select “Color Fill” as the *Contour Method*.

5. Under the *Vectors* tab in the *Arrow Options* section, change the *Shaft Length* option to "Scale length to magnitude."
6. Change the scaling *Ratio* to "4.0."
7. When done, close the *Display Options* dialog by clicking **OK**. The mesh should appear similar to Figure 1.

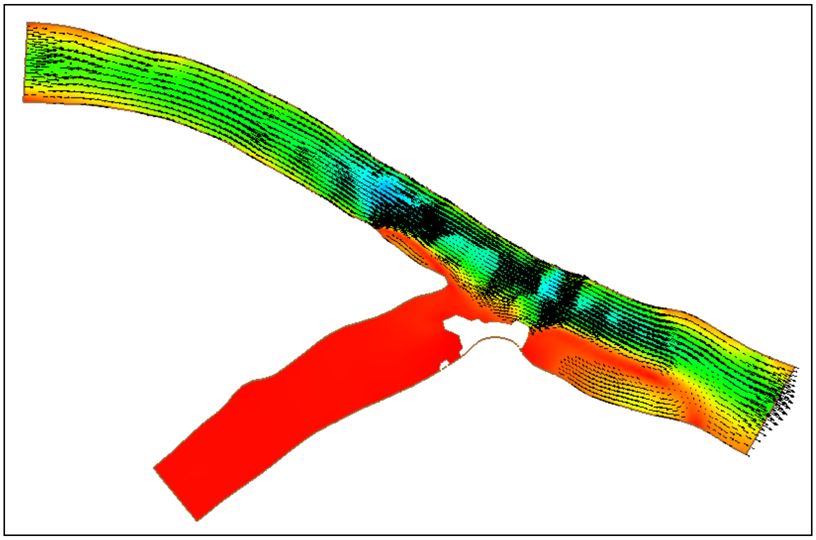


Figure 1 The mesh of the velocity dataset with contours and vectors turned on

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:

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

Do as follows to create the Froude number dataset:

1. Select *Data* | **Dataset Toolbox** to bring up the *Dataset Toolbox* dialog. The *Data Calculator* is contained within this dialog.
2. Select "Data Calculator" from under the *Tools* section.
3. Highlight the "d2. velocity mag" dataset under the *Datasets* section.
4. Under the *Time Steps* section, turn on the *Use all time steps* option.
5. Click the **Add to Expression** button. The Calculator will show "d2:all."

   Note: "d2" corresponds to the "velocity mag" dataset and "all" signifies all time steps.
6. Click the **divide** "/" button.

7. Click the **sqrt** button.

   Note: The "??" text is just a placeholder to indicate that something should be placed there. It should be highlighted.

8. Enter "32.2" to replace "??" for the constant g.
9. Click the **multiply** "*" button.
10. Highlight the "d3. water depth" dataset.
11. Click the **Add to Expression** button.
12. The expression 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 could also just be typed in directly.)
13. In the *Output dataset name* field, enter the name "Froude".
14. Click the **Compute** button. SMS will take a few moments to perform the computations. When it is done, the "Froude" dataset will appear in the *Datasets* window.
15. Click the **Done** button to exit the *Dataset Toolbox* dialog.

The "Froude" dataset can be contoured and edited with any of the other tools in SMS, just as any other dynamic scalar dataset. It can 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 example, create contours for the velocity magnitude dataset by doing the following:

1. Switch to the velocity magnitude dataset by choosing "velocity mag" in the Project Explorer.
2. Click on "0 00:00:00" in the *Time steps* list box below the Project Explorer.
3. Click on the **Display Options** macro to bring up the *Display Options* dialog.
4. Select *2D Mesh* from the list on the left then click the **All Off** button to turn off all existing display options.
5. Turn on the following options:
   - *Contours*
   - *Mesh boundary*
   - *Wet/dry boundary*

6. Select the *Contours* tab and set the *Contour Method* to “Linear”.
7. The *Contour interval* should be set to “Number” and enter “20” in the field to the right.
8. Click **OK** to close the *Display Options* dialog. The “velocity mag” dataset should appear similar to Figure 2.

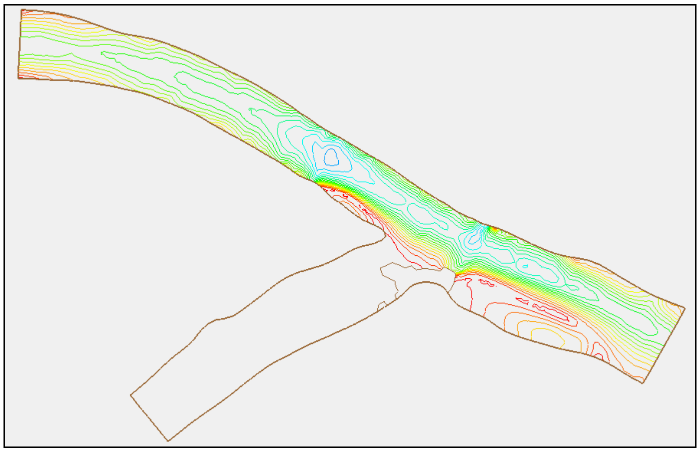


Figure 2 Linear contour display

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

1. Right-click on the “Mesh Data” folder in the Project Explorer and select the **Display Options** command (this is an alternative to using the macro). This will bring up the *Display Options* dialog.
2. The *2D Mesh* in the list on the left should be highlighted. Switch to the *Contours* tab.
3. Change the *Contour Method* to “Color Fill.”
4. Make sure the *Fill continuous color range* option (located at the bottom right side of the dialog) is on. This option causes SMS to blend dataset values rather than use discreet intervals.
5. Click **OK** to close the *Display Options* dialog and see color filled contours on the mesh.

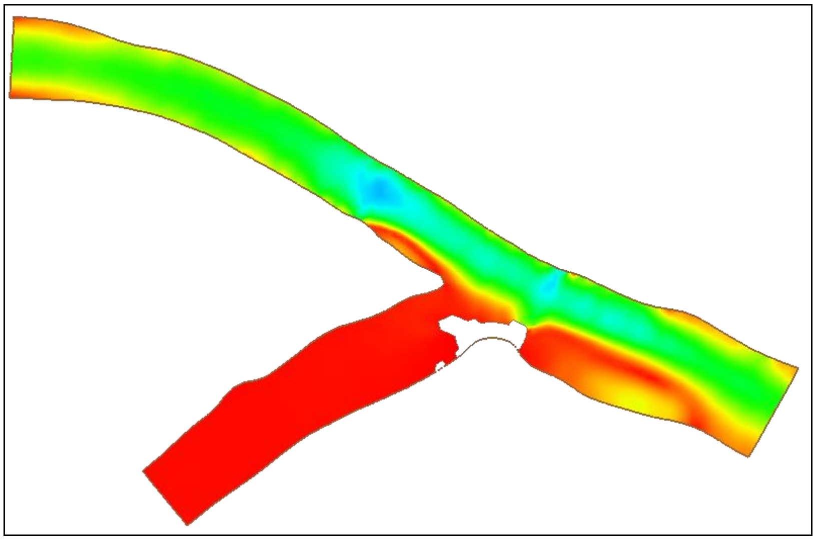


Figure 3 Color filled contour display

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. Switch to the water depth dataset by choosing “water depth” in the Project Explorer.
2. Click on the **Display Options** button to bring up the *Display Options* dialog.
3. With *2D Mesh* highlighted in the list on the left, select the *Contours* tab.
4. Click on the **Color Ramp** button. This will bring up the *Color Options* dialog.
5. Select the *User defined* radio button.
6. Click the **New Palette** button. This will bring up the *New Palette* dialog.
7. Change the *Initial Color Ramp* to “Ocean.”
8. Click **OK** to exit the *New Pallette* 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.

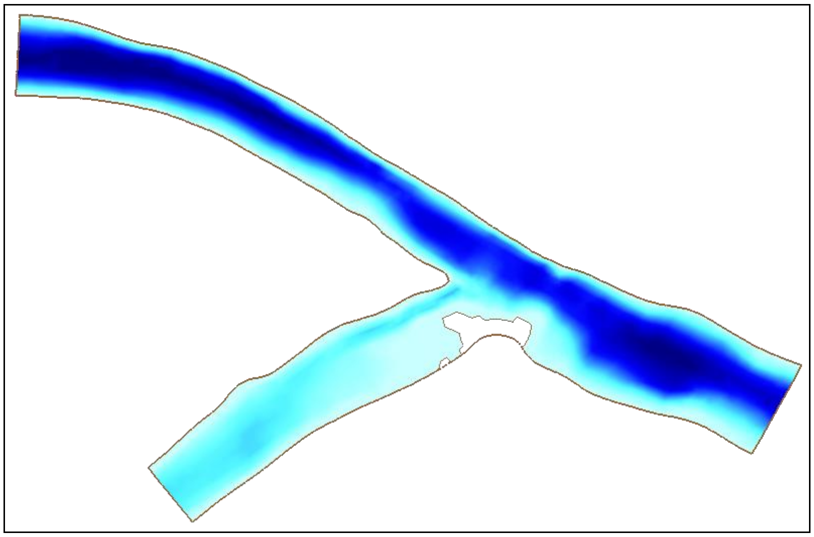


Figure 4 Using the color ramp with the ocean color palette

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. Switch to the velocity magnitude dataset by choosing “velocity mag” in the Project Explorer.
2. Click on the **Display Options** button to bring up the *Display Options* dialog.
3. With *2D Mesh* highlight, turn on the *Vectors* option.
4. Switch to the *Contours* tab.
5. Click on the **Color Ramp** button to bring up the *Color Options* dialog.
6. In the *Pallette Method* section, select *Hue ramp*.
7. Click on the **Reverse** button at the bottom of the dialog to make red indicate the higher velocities.
8. Click **OK** to exit the *Color Options* dialog.
9. Select the *Vectors* tab.
10. In the *Arrow Options* section, set *Shaft Length* to “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.
11. Enter “10” in the *Minimum* field.
12. Enter “80” in the *Maximum* field.

13. 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. To thin out the arrows, follow these steps:

1. Click on the **Display Options** button to bring up the *Display Options* dialog.
2. With *2D Mesh* highlighted in the list on the left, select the *Vectors* tab.
3. In the *Vector Display Placement and Filter* section, find *Display* and choose "on a grid."
4. Enter "25" in both the *X spacing* and *Y spacing* edit fields.
5. Enter an O*ffset* of "5.0".
6. Click **OK** to exit the *Display Options* dialog.

Now the arrows should be evenly distributed over the domain at 25 pixel increments. 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.

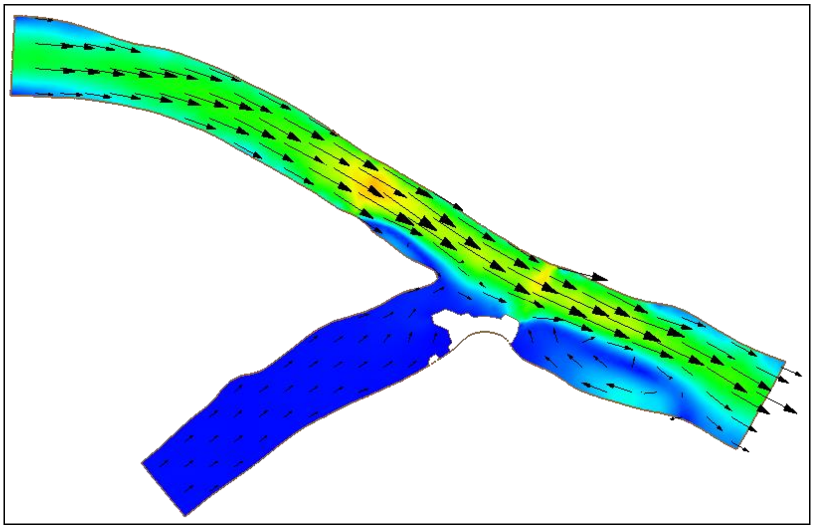


Figure 5 Vector display

Right below where the two branches join, an eddy is formed.

7. **Zoom** in around the eddy as shown in Figure 6.

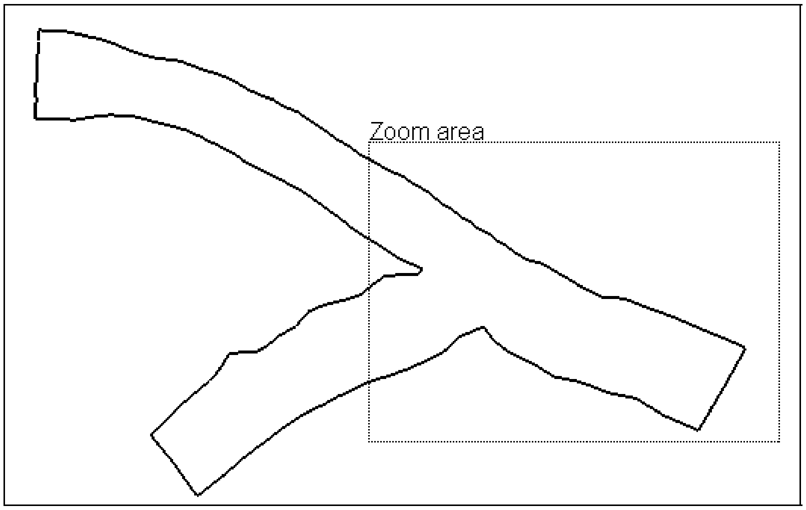


Figure 6 Area to zoom to

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

8. When done click the **Frame** macro.

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, follow these steps:

1. Hold down the *Shift* key to select both the "velocity mag" (scalar) and "velocity" (vector) datasets in the Project Explorer.
2. Select *Data* | **Film Loop** to bring up the *Film Loop Setup – General Options* dialog.
3. Make sure that *Create AVI File* is selected under *Film Loop Files*.
4. Make sure *Transient Data Animation* is selected for the *Select Film Loop Type*.
5. Click the **File Browser** button. A *Save* dialog will appear.
6. In the *Save* dialog, enter the name "velocity.avi."
7. Click **Save** to go back to the *Film Loop Setup—General Options* dialog.
8. Click **Next** to go to the *Film Loop Setup—Time Options* dialog.

9. It is necessary to set up the time step and time length for the animation. The default settings match the solution time step and duration for this tutorial. Click **Nex**t to go to the *Film Loop Setup—Display Options* dialog.
10. The last page of the setup allows the display options to be modified and a clock to be specified. Click on the **Clock Options** button to bring up the *Clock Options* dialog.
11. Change the *Location* to the “Top Right Corner.”
12. Click **OK** to close the *Clock Options* dialog.
13. Click the **Finish** button to close the *Film Loop Setup* wizard and to create the film loop.

SMS will display each frame of the film loop as it is being created. When the film loop has been fully generated, it will launch in a new *Play AVI Application* (PAVIA) window. This application contains the following controls:

- ***Play*** button. This starts the playback animation. During the animation, the speed and play mode can be changed.
- ***Speed*** slider bar. This increases or decreases the playback speed. The speed depends on the computer being used.
- ***Frame*** slider bar. This control can be used to jump to a specific frame of an animation.
- ***Stop*** button. This stops the playback animation.
- ***Step*** button. This allows manually stepping to the next frame. It only works when the animation is stopped.
- ***Loop*** play mode. This play mode restarts the animation when the end of the film loop has been reached.
- ***Back/forth*** play mode. This play mode shows the film loop 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 the PAVIA application. Pavia.exe is located in the SMS installation directory and can be freely distributed.

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. Close the *Play AVI Application* window.
2. Click the **Display Options** button to bring up the *Display Options* dialog.
3. With *2D Mesh* highlighted, uncheck the *Vectors* option and check on the *Functional Surface* option.
4. Click on the **Options** button to the right of the *Functional Surface* option to open the *Functional Surface Options* dialog.
5. In the *Dataset* section, select *User defined dataset*. A *Select Dataset* dialog will appear.
6. Choose the "water surface elevation" dataset. Then click **Select** to close the dialog.
7. Click **OK** to close the *Functional Surface Options* dialog.
8. Switch to the *General* option in the list on the left of the *Display Options* dialog.
9. Turn off *Auto Z-mag*.
10. Change the *Z magnification* under *Drawing Options* to "5.0."
11. Click **OK** to close the *Display Options* dialog.
12. Select the *Display* | *View* | **Oblique** command to change the view to an oblique (3D) view.
13. Open the *Display Options* dialog again.
14. Make certain the *General* page is active and switch to the *View* tab.
15. Under the *View angle* section enter a *Bearing* of "43."
16. Enter a *Dip* of "22."
17. For *Looking at Point* enter the values of "17275," "13900," and "5.75" for *X, Y,* and *Z*, respectively.
18. Under *Defined view bounds size,* enter a *Width* of "3200."
19. 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 7.

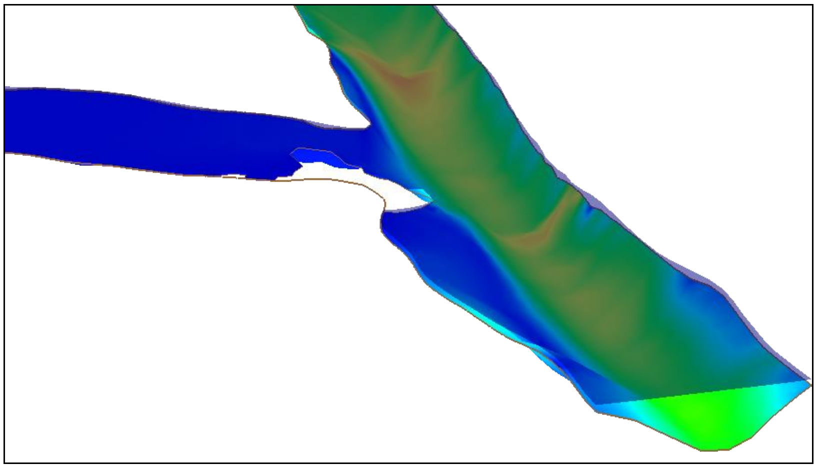


Figure 7 Functional surface

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.

20. To view the animation, follow the steps from section 6.1 to animate the functional surface. Name the animation “wse.avi.”

The selected values are to illustrate the oblique view.

21. Select any view using the **Rotate** tool.
22. When done, click the **Frame** macro to view the entire model.

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.

Note that a flow trace generally takes longer to generate than the scalar/vector animation. 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. Click on the **Plan View** button.
2. **Zoom** in on the area of the junction of the two reaches.
3. Select *Data* | **Film Loop** to bring up the *Film Loop Setup* wizard.
4. In the *Select Film Loop Type* section, select the *Flow Trace* option.
5. Click the **File Browser** button. A *Save* dialog will appear.
6. Change the file name to “flowtrace.avi” then click **Save**.

7. Then click the **Next** button.
8. Use the default time settings again by clicking **Next**.
9. Enter “0.5” as the number of *Particles per object*.
10. Enter “0.1” as the *Decay ratio*.
11. Leave all other options in the *Flow Trace Options* page as their default values and click the **Next** button.
12. Click the **Finish** button to generate the animation.

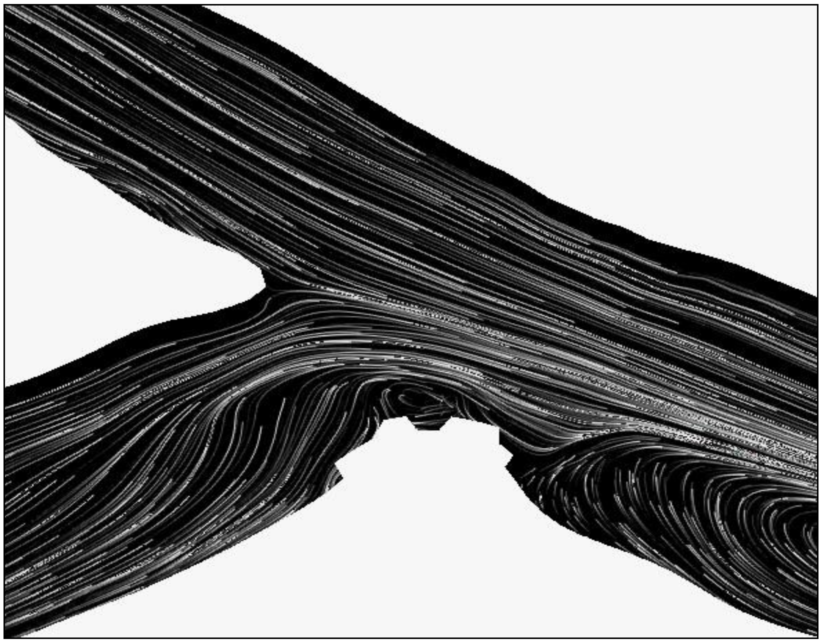


Figure 8 One frame from the ld1 flow trace

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 the previous animation was. The flow trace can be viewed using the same controls available for the other film loop animation.

13. When done, close the *Play AVI Application* window.

6.4 Drogue Plot Animation

Drogue plot animations are similar to flow trace animations, except that they specifying 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 9.

3. Click on the **Display Options** command to open the *Display Options* dialog.
4. Turn off the *Functional surface* option then click **OK**.
5. Click on the “Area Property” coverage in the Project Explorer to switch to the Map module.
6. Right-click on the “Area Property” item and select the *Type* | *Generic* | **Particle/Drogue** command.
7. Select the **Create Feature Arcs** tool.
8. Create a feature arc across one branch of the river (as shown in Figure 9) by clicking on one side of the branch, clicking to make vertices while crossing the branch, and then double-clicking on the opposite bank to complete the arch. Repeat the process for the other branch.
9. Select both arcs with the **Select Feature Arcs** tool by holding down the *Shift* key while clicking on each arc.
10. Choose *Feature Objects* | **Redistribute Vertices** to bring up close the *Redistribute Vertices* dialog.
11. Change the *Specify* option to “Number of Segments.”
12. Set the *Number of* to “20.”
13. Click **OK** to close the *Redistribute Vertices* dialog.
14. Create three individual points in the downstream branch of the river with the **Create Points** tool as shown in Figure 9.

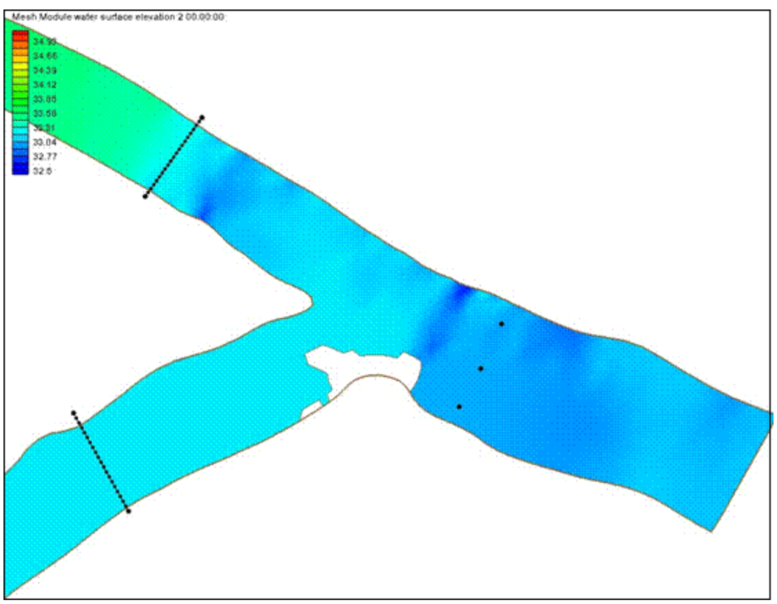


Figure 9 Feature objects for the particle/drogue coverage

The arcs and points that were defined should look something like Figure 9. For the drogue plot animation, one particle will be created at each feature point and each feature arc vertex.

To create the drogue plot animation:

1. Click on the Mesh module in the Module Toolbar to activate it.
2. Select *Data* | **Film Loop** to bring up the *Film Loop Setup* wizard again.
3. Select the *Drogue Plot* animation type.
4. Click the **File Browser** button. A *Save* dialog will appear.
5. Change the *File name* to "drogue.avi." then click **Save**.
6. Then click **Next** to go to the *Time Options* page. (The coverage was just created, so it is already set.)
7. In the *Filmloop Time* section, change the *Run Simulation For* to "5.0" hours.
8. Click on the *Specify Time Step Size* radio button.
9. Specify a time step size of "5.0" minutes (make sure to change the drop-down menu from "hours" to "minutes").
10. Click **Next** to go to the *Drogue Plot Options* page.
11. In the *Color Options* section, click the *Distance traveled* radio button.
12. Set the *Maximum value* to "1500."
13. Turn on the *Write report* option.
14. Click **Next** then click the **Finish** button to generate the animation.

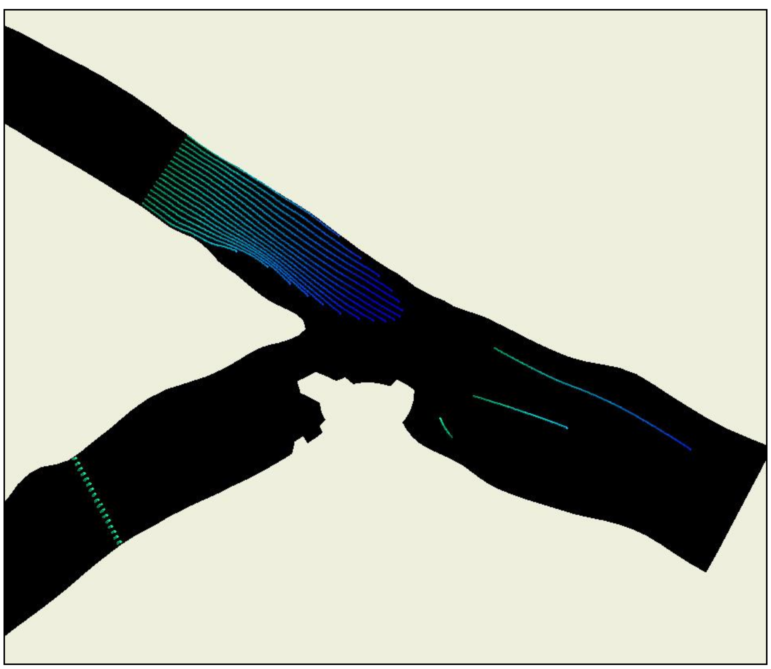


Figure 10 Sample drogue plot animation

The drogue plot animation generates a report by turning on the option in step number 3 above. To see this report, do the following:

1. Close the *Play AVI Application* window.
2. Choose *File* | **View Data File** to bring up an *Open* browser.
3. Select the file “drogues.pdr” in the *data files* folder for this directory or in wherever the drogue animation was saved. Click **Open.**
4. A *View Data File* dialog may appear asking which application to use to open the file. After selecting the application from the *Open with* dropdown menu, either turn on *Never ask this again* and then click the **OK** button, or just click the **OK** button.

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. Continue to experiment with the SMS interface or quit the program.

