SURFACE WATER MODELING SYSTEM

Data Visualization

1 Introduction

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. In this lesson, you will learn how to import, manipulate and view solution data. You will need the geometry file *dv_001.2dm* and the solution file *dv_001.h5.* These files are in the Data Files Folder for this tutorial.

2 Data sets

A geospatial data set has one or more numeric values associated with each node in a mesh, cell in a grid, vertex in a scatter set, etc. Scalar data sets have one value per location. Two-dimensional vector data sets have two values for every location (an x-component and a y-component). Examples of scalar data sets include bathymetry, water surface elevation, velocity magnitude, Froude number, energy head, concentration, bed change, wave heights and many more. Examples of vector data sets include observed wind fields, flow velocities, shear stresses, and wave radiation stress gradients.

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

3 Open the Geometry and Solution Files

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

1. Select *File* | *Open*.

2. Open the file "dv_001.2dm" from the Data Files Folder for this tutorial.

With the geometry opened, the solution can be imported. To import the solution file:

1. Select *File* | *Open*.

2. Open the file "dv_001.h5" from the Data Files Folder. This is a file generated by *SMS* which includes data sets for water depths, water surface elevations, velocity magnitudes and velocities.

SMS displays the data sets as contours and vectors. To be consistent:

1. Open the *Display Options* dialog.

2. Under the *2D Mesh* tab, check the *Contours* and *Vectors* options. Also, turn off the *Nodes* option.

3. Under the *Contours* tab, select *Color Fill* as the *Contour Method*.

4. Under the *Vectors* tab, change the *Shaft Length* option to *Scale length to magnitude.* Change the scaling ratio to 4.0. Close the *Display Options* dialog by clicking *OK*.

4 Creating New Data sets with the Data Calculator

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

$$\text{Froude Number} = \frac{\text{Velocity Magnitude}}{\sqrt{\text{gravity} \ * \ \text{water depth}}}$$

To create the Froude number data set:

1. Select *Data* | *Data Calculator*.

2. Highlight the *velocity mag* data set.

3. Under the *Time Steps* section, turn on the *Use all time steps* option and click the *Add to Expression* button. The Expression will show "d2:all". The letter 'd' corresponds to the *velocity mag* data set and 'all' signifies all time steps

4. Click the *divide* "/ " button.

5. Click the sqrt(x) operation. The “??” text is just a placeholder to make sure you know that something should be placed there. It should be highlighted. Enter 32.2 for the constant g.

6. Click the *multiply” * “* button, then highlight the *water depth* data set and click the *Add to Expression* button.

7. The expression should now read: “d2:all / sqrt(32.2 * d3:all)”, where ‘d2’ represents the velocity data set and ‘d3’ represents the water depth data set. (This expression could also just be typed in directly.)

8. In the Output dataset name field, enter the name *Froude* and then click the *Compute* button. *SMS* will take a few moments to perform the computations. When it is done, the *Froude* data set will appear in the *Data Sets* window.

9. Click the *Done* button to exit the *Data Calculator* dialog.

The Froude data set is automatically placed in the dv_001.sol folder by default.

This data set can be contoured and edited with any of the other tools in *SMS*. It can be treated just as any other dynamic scalar data set and can be saved in a generic data set file. See the *SMS Help* for more information on saving data sets.

5 Contours

Turning on Contours

SMS provides several contour options to help visualize data sets. For this example, we will create contours for the velocity magnitude data set. To create contours for the velocity magnitude data set:

1. Switch to the velocity magnitude data set by choosing v*elocity mag* in the *Project Explorer*. Click on *0 00:00:00* in the Time Step list-box below the *Project Explorer*.

2. Click on the *Display Options* macro .

3. Click *All off* to turn off all existing display options.

4. Turn on the *Mesh boundary*, the *Wet/dry boundary* and *Contours*.

5. Select the *Contours* tab.

6. Set the *Contour Method* to *Linear* and the *Number of Contours* to 20.

7. Click *OK*.

In addition to linear contours, *SMS* also supports color-filled contours as well as color-filled with linear contours at the breaks. To use color-filled contours:

1. Right click on the "Mesh Data" item in the *Project Explorer* and select the *Display Options* command (this is an alternative to using the macro).

2. 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 toggle causes SMS to blend data set values rather than use discreet intervals.

5. Click *OK* to see color-filled contours on the mesh.

5.2 Color Ramp Options

The default color ramp in *SMS* has dark blue for the largest scalar value to 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 user's default when running *SMS*. To use a different color-ramp to better visualize water depths:

1. Switch to the water depth data set by choosing water depth in the *Project Explorer*.

2. Bring up the *Display Options* (by either method already used).

3. Select the *Contours* tab.

4. Click on the *Color Ramp* button.

5. Select the *User defined* radio button.

6. Click the *New Palette* button.

7. Change the *Initial Color Ramp* to *Ocean*.

8. Click *OK* three times to get back to the main SMS screen.

This color ramp shows the deeper areas as dark blue and shallower areas as light blue.

6 Vectors

Vector data sets can be visualized inside of *SMS* by displaying arrows representing the direction and optionally the magnitude of the vector data set over the mesh.

To turn on vectors for the velocity data set:

1. Switch to the velocity magnitude data set by choosing *velocity mag* in the *Project Explorer*.

2. Bring up the *Display Options*.

3. Click the toggle labeled *Vectors* on the *2D Mesh* tab.

4. Switch to the *Contours* tab.

5. Click on the *Color Ramp* button and change back to a *Hue ramp*. Click on the *Reverse* button at the bottom of the dialog to make red indicate the higher velocities. Click *OK*.

6. Select the *Vectors* tab.

7. In the *Shaft Length* section choose *Define min and max length*. This scales the length of the arrows based upon the magnitude of the velocity data set at the arrow location. The minimum data set magnitude uses the shaft length that is the minimum length. Likewise the maximum data set magnitude uses the maximum shaft length. Enter values of 10 and 80 in the two fields.

8. Click *OK*.

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:

1. Bring up the *Display Options* and click on the *Vectors* tab.

2. In the *Vector Display Placement and Filter* section, find *Display* and choose "on a grid" and enter 25 in both of the "pix" edit fields. Enter a Z-offset of 5.0 and click *OK*.

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.

Right below where the two branches join, an eddy is formed. Zoom in around the eddy as shown in Figure 1.

Figure 1 Area to zoom to.

As you zoom, the vector spacing stays at 25 pixels. Therefore, additional vectors appear illustrating the recirculation pattern.

7 Creating Animations

A film loop is an animation created by *SMS* to display changes in data sets through time. Flow trace and particle trace animations are a special type of film loop, which use vector data sets 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.

7.1 Creating a Film Loop Animation

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

1. Make sure that the *velocity mag* (scalar) and *velocity* (vector) data sets are active by selecting them in the *Project Explorer.*

2. Select *Data* | *Film Loop*.

3. Make sure *Transient Data Animation* is selected for the *Film Loop Type*.

4. Make sure that *Create AVI File* is selected under *Film Loop Files* and click the *File Browser* button and enter the name "velocity.avi" and click *Save*. Then click *Next*.

5. In the *Time Options* page, we need to setup the time step and time length for the animation. The default is to match the solution time step and duration, which we want to use. Click Next.

6. The last page of the setup allows the display options to be modified and a clock to be specified. Click on the *Clock Options* and change the "Location" to the "Top Right Corner". Click the *OK* button and then click the *Finish* button in the *Film Loop Setup* wizard 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:

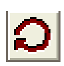
- *Play*  *button*. This starts the playback animation. During the animation, the speed and play mode can be changed.

- *Speed*. This increases or decreases the playback speed. The speed depends on your computer.

- *Frame*. 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 you to manually step 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.

7.2 Animating a Functional Surface

Functional surfaces can be used to visualize data sets as a surface with the elevation at each node being the value of the data set plus a constant offset. A functional surface can be used to display the water surface. To turn on the functional surface:

1. Bring up the *Display Options* dialog.

3. Turn off the vectors option.

4. Click on the *Functional Surface* toggle.

5. Click on the *Options* button to the right of the *Functional Surface* option.

6. In the *Data Set* section, select "User defined data set". In the *Select Dataset* dialog, choose the "water surface elevation" data set. Then click *Select*.

7. Click *OK* in the *Functional Surface Options* dialog.

8. Switch to the *General* tab.

9. Toggle off the Auto Z-mag.

10. Change the *Z magnification* under drawing options to 5.0.

11. Click *OK*.

12. Select the *Display | View | View Options* command to change the view to an oblique (3D) view. Under "View angle", enter a *Bearing* of 43, and *Dip* of 22, a *Look At Point* of (17275, 13900, 575) and a *Width* of 3200. Then click *OK*. (These values are selected to illustrate the oblique view, you can select any view you want using the rotate tool .)

The functional surface of water surface should appear over the bathymetry, shaded with the 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. To view the animation:

- Follow the steps from the previous section to animate the functional surface. Name the animation as *wse.avi*.

7.3 Creating a Flow Trace Animation

A flow trace animation can be created if a vector data set 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 you have problems with this operation, decrease the size of the *SMS* window and try again. To create and run a flow trace film loop:

1. Click on the *Plan View* button.

2. Zoom in on the area from the junction of the two reaches.

3. Select *Data* | *Film Loop*.

4. In the *Select Film Loop Type* section, select the *Flow Trace* option and change the file name to flowtrace.avi. Then click the *Next* button.

5. Again we want to use the default time settings. Click *Next*.

6. Enter 0.5 as the number of *Particles per object* and 0.1 as the *Decay ratio*. Leave all other options in the *Flow Trace Options* page as their default values and click the *Next* button.

7. Click the *Finish* button to generate the animation.

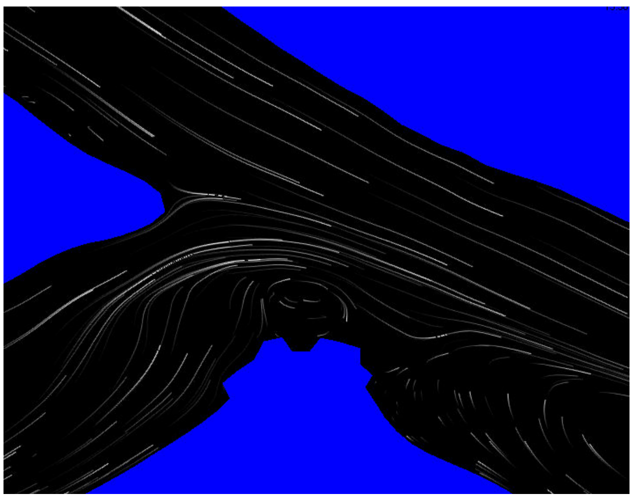


Figure 2. One frame from the ld1 flow trace

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 a prompt shows which frame is being created. When the flow trace has been created, it is launched in a new window, just as the previous animation.

- View the flow trace using the same controls as with the film loop animation.

7.4 Drogue Plot Animation

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

1. Zoom into the area shown in Figure 3.

2. Turn off the functional surface in the *Display Options* then press OK.

3. Click on the “default coverage” item in the *Project Explorer* to switch to the Map module. Then right click on the “default coverage” item and select the “*Type*” command. Then select “*Generic*” then *Particle / Drogue*.

4. Create two feature arcs with the *Create Feature Arcs* tool, one across each upstream branch of the river.

5. Select both arcs with the *Select Feature Arcs* tool by holding *SHIFT* while clicking on each, and choose *Feature Objects* | *Redistribute Vertices*.

6. Change the *Specify* option to *Number of Segments* and set *Num Seg* to 20.

7. Click *OK* to close the *Redistribute Vertices* dialog.

8. Create three individual points in the downstream branch of the river with the *Create Points* tool.

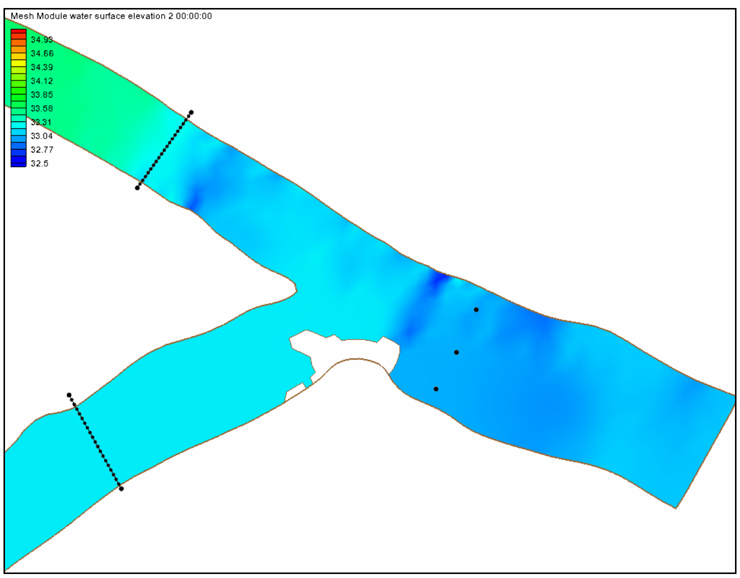


Figure 3. Feature objects for the particle/drogue coverage.

The arcs and points that were defined should look something like Figure 3. 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” item to switch back to the *Mesh* module and select *Data* | *Film Loop*.

2. Select the *Drogue Plot* animation type and change the filename to “drogue.avi”. Then click the *Next* button (the coverage was just created so it is already set).

3. In the Time Step section, Change the Run Simulation For to 5.0 hours. Specify a time step size of 5.0 minutes. Click *Next*.

4. In the *Color Options* section, associate the color ramp with the *Distance traveled*, and set the *Maximum distance* to 1500. Turn on the *Write report* option, then click *Next*.

5. Click the *Finish* button to generate the animation.

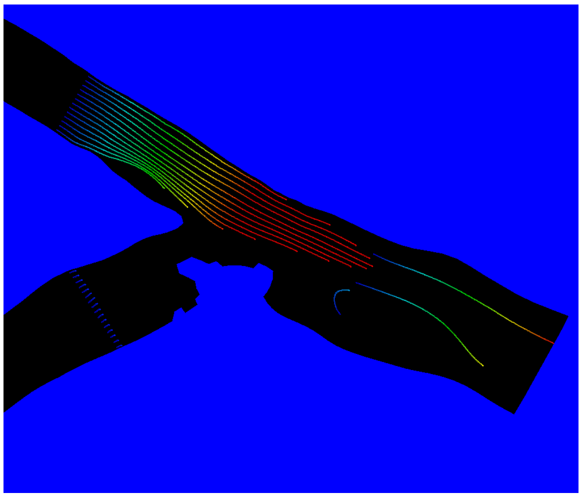


Figure 4. Sample drogue plot animation.

The drogue plot animation generates a report by turning on the option in step number 4 above. To see this report:

- Choose *File* | *View Data File* and Open the file "drogues.pdr".

Now that you have seen the three main animation types available in SMS 10.0, feel free to experiment with some of the available options, especially with the flow trace and drogue plot animation types.

8 2D Plots

Plots can be created to help visualize the data. Plots are created using the observation coverage in the map module. Lesson 4 teaches how to use the observation coverage.

9 Conclusion

This concludes the Data Visualization tutorial. You may continue to experiment with the *SMS* interface or you may quit the program.