

---

## Visualization

### 5.1 Introduction

---

This workshop will teach you how to use the visualization tools that are available in SMS. These tools can help you analyze the modeling results, a procedure that is often called ‘post-processing’. For this session, you will need the following files:

- Shinnecock.grd
- Fort.63
- Fort.64

### 5.2 Opening the Solution

---

Solution files are opened into SMS just like any other file, but before you can open a solution file, the corresponding geometry file must first be opened. To begin this workshop:

1. Select *File | Open* and open the file “shinnecock.grd”. This model is of Shinnecock Inlet and surrounding areas around Long Island, New York. The geometry is shown in Figure 5-1.

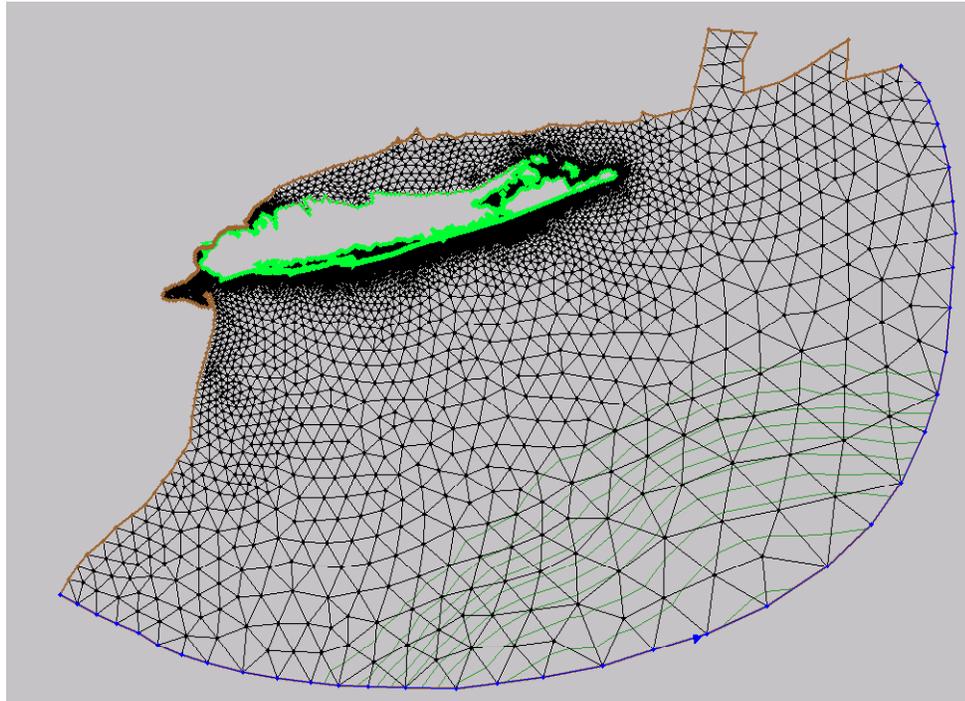


Figure 5-1. The Shinnecock model geometry

2. Open the file “fort.63”. This file contains the time variant water surface elevations computed by *ADCIRC*.

### 5.3 Data Sets

---

When you open a solution file into *SMS*, a number of *data sets* are created, one for each quantity in the file. Data sets are either scalars, with one value per object, or vectors, with two values per object. A finite element data set contains solution data at each node of the mesh.

*SMS* can open a number solution file types. Table 5-1 shows the supported types. The type of data that is available depends upon the model. For example the *ADCIRC* hydrodynamics files used in this lesson contain water surface elevation and velocity information.

Table 5-1 Supported Finite Element Solution types.

File Type	Source of File Type
Generic file	HIVEL2D, SMS-created data files, Other sources
TABS file	RMA2, RMA4, SED2D-WES, RMA10
FESWMS file	FLOMOD, FLO2DH
ADCIRC file	ADCIRC hydrodynamics
ADCIRC harmonic file	ADCIRC wave information
CGWAVE file	CGWAVE

Data sets have one or more time steps. A data set that has only one time step is steady state. Transient data sets have multiple time steps. Data sets are grouped together into “solutions” in *SMS*. When opening a file, all data sets in the file are read in as a new solution or added to the current solution. Other data sets may be added this solution or data sets may be moved out of the solution. Generally speaking, all data sets in a single solution usually have the same number of time steps.

## 5.4 Using the Data Browser

Data sets and solutions are managed through the *Data Browser*. To open this dialog:

- Select *Data* | *Data Browser*.

The data browser should look like Figure 5-2.

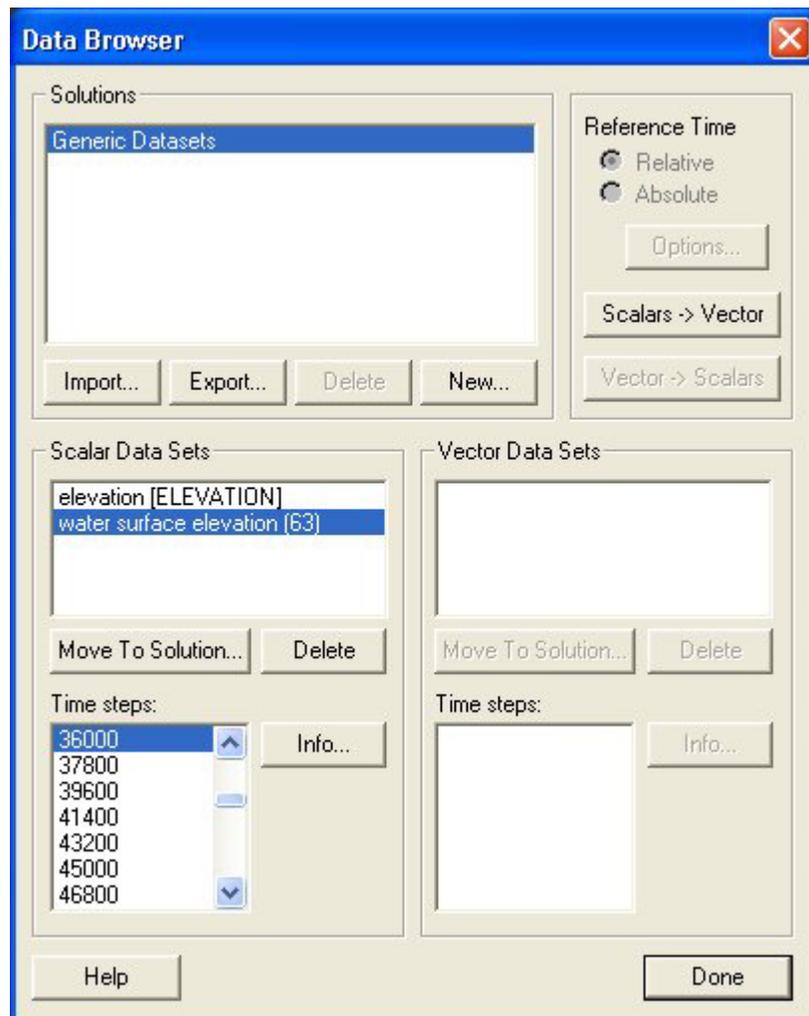


Figure 5-2. The Data Browser.

Notice the following sections in the data browser:

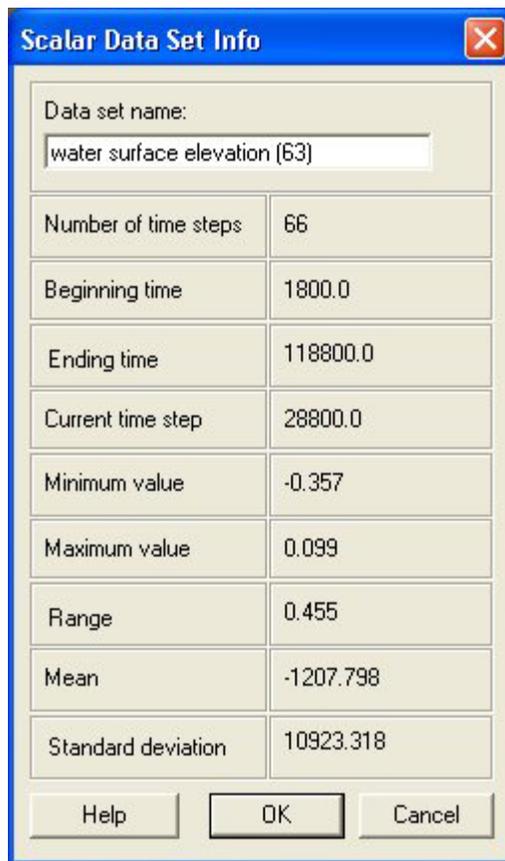
- *Solutions*. This section shows all the solutions that have been opened. There is also a solution named “Generic Datasets”. This solution is a sort of catch-all solution and contains datasets such as node depths (bathymetry). The highlighted solution becomes the active solution.
- *Scalar Data Sets*. This section shows the scalar data sets that exist in the active solution. The highlighted one is the active scalar. If the solution is transient, a set of time steps will also be shown. Contour plots use the active scalar data set.
- *Vector Data Sets*. This section shows the vector data sets that exist in the active solution. The highlighted one is the active vector. If the solution is transient, a set of time steps will also be shown. Vector plots use the active vector data set. In this example, no vector sets have been read in yet so the list is empty.
- *Command Buttons*. There is a set of commands that can be executed by clicking the appropriate button.

Notice that the water surface elevation was placed by default inside the Generic Datasets solution. We want it to be a new solution created for this particular run. To move this dataset to a new solution:

1. Click on the button *New* right below the solution window.
2. Enter the solution name *Shinnecock*.
3. Select the *Generic Datasets* solution to make it active and select the data set named *water surface elevation (63)*.
4. Click on the *Move To Solution* button below the scalar data sets window. A small dialog appears with a combo box to select the solution to move this data set to. Select the *Shinnecock* solution and click *OK*.
5. Select the *Shinnecock* solution in the data browser to make it active again and you will see the *water surface elevation (63)* data set in this solution.

You can get statistical information about a single time step of a data set by highlighting it and clicking the appropriate *Info* button (scalar or vector version). To check the statistical info for the current water surface elevation function:

1. Highlight the function named “water surface elevation”.
2. Click the *Info* button to open the *Scalar Data Set Info* dialog, as shown in Figure 5-3. In transient simulations, the statistical information (min, max, range, mean, deviation) is computed from only the active time step.
3. Click the *OK* button to close the *Info* dialog.



The dialog box titled "Scalar Data Set Info" displays the following information:

Data set name:	water surface elevation (63)
Number of time steps	66
Beginning time	1800.0
Ending time	118800.0
Current time step	28800.0
Minimum value	-0.357
Maximum value	0.099
Range	0.455
Mean	-1207.798
Standard deviation	10923.318

Buttons: Help, OK, Cancel

Figure 5-3. Data set statistical information.

We want to open the velocity solution from *ADCIRC*. We can do this from the file menu, or from inside the Data Browser. To load the velocity data computed by *ADCIRC* into the solution we just created:

1. Click on the button *Import*.
2. Change the file type to *ADCIRC Unit 64* and select the file *fort.64*.
3. Choose *Add to solution set* and select the solution set *Shinnecock*.
4. Click *OK* and wait for *SMS* to read in the velocity file.
5. Select the *water surface elevation function* to make it the current function.
6. Scroll the time step window and select time 97,200 (this is measured in seconds and represents 27 hours from the start of simulation time and corresponds to an ebb tide period).
7. Click *Done* to close the *Data Browser*.

After closing the *Data Browser* dialog, the *SMS* display will refresh. Contours in the new display will correspond to the water surface elevation 24 hours after the start time of the simulation.

Note: *ADCIRC* solution files can be very large (gigabytes) when full lunar cycles have been simulated. These files may take several minutes to read into *SMS*.

### 5.5 The SMS Edit Bar

---

The *SMS* Edit Bar lies just below the menus in the *SMS* window. It is shown in Figure 5-4. Among other things, this window has drop-down lists that can be used as short-cuts to set the active solution, data set, and time step.



Figure 5-4. The *SMS* Edit Bar.

To change the active scalar data set back to the mesh elevation (bathymetry):

- In the *Solution* drop-down list, choose “Generic Datasets”. Right now, the only data set in this solution list will be elevation, so it becomes active.

### 5.6 Contour Plots

---

Now let’s look at how data sets are used for visualization. First, we will look at contour plots. In *SMS*, the user can control many options such as how many contours are displayed, what values they are associated with, what type of contour is displayed (line or solid color), and the thickness of contour lines. To set up the display options for a new contour plot:

1. Click the *Display Options*  macro.
2. In the *2D Mesh* tab, turn off all options except for *Contours* and *Mesh boundary*.
3. Click on the *Contours* tab to switch to the contour display options.
4. Click on the *Color Options* button. This will bring up a color options dialog.
5. Select *Hue Ramp* and click *OK*. Now each bathymetric depth contour will have its own color instead of a constant green. (After installation, the default type of contours is green linear contours. You can change the default type by saving a “Settings file”.)
6. Click *OK* to close the *Display Options* dialog

You should see an image similar to Figure 5-5. The contours are all out to sea showing the increased depth in that region. The contours you should see now show the active data set which should still be the bathymetry.

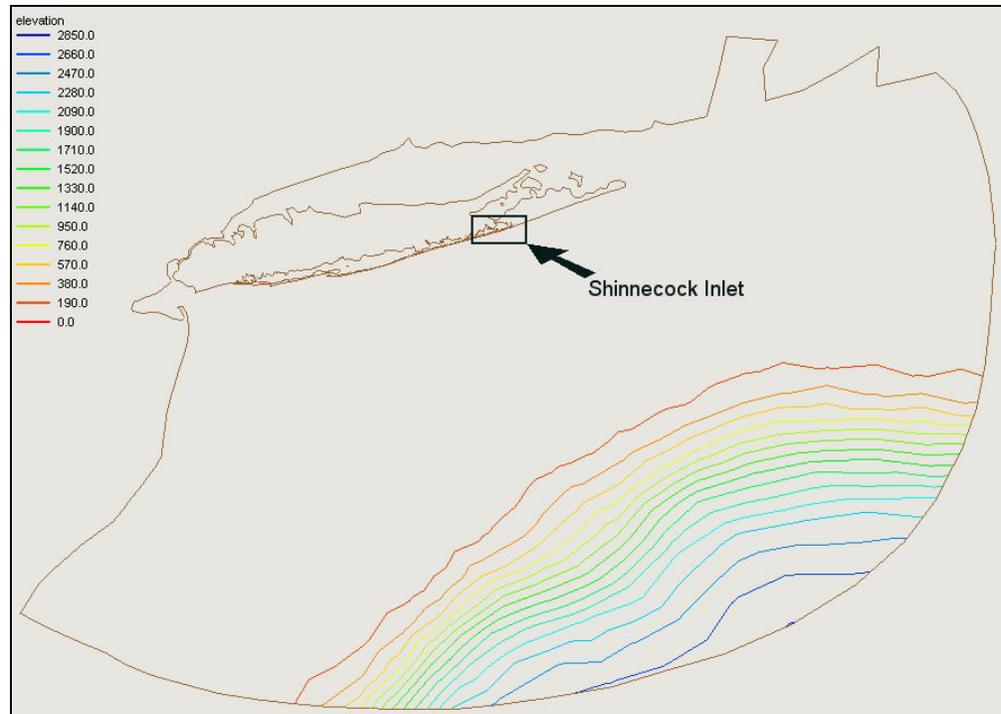


Figure 5-5. The Shinnecock model geometry

Zoom in on Shinnecock Inlet as shown in Figure 5-5.

To view some solution data:

- Change the active solution back to Shinnecock and then set the active scalar to magnitude. (The timestep will still be set to 97200 or 27 hours into the simulation).

### 5.6.1 Changing Contour intervals

This plot isn't very informative because for this time step, none of the velocity magnitudes are above 1.3 m/s and most are less than 0.5 m/s. SMS chooses a default contour range based on the extents of the entire data set (all time steps). For this view, with this time step, the distribution isn't very good. Let's adjust our contour settings to visualize the velocity distribution better. To do this:

1. Click the *Contour Options*  macro. (This identical to opening the display options dialog and selecting the *Contours* tab.)
2. In the contour range box, select *specify a range*.

3. Set the *minimum value to 0.0* and the *maximum value to 2.0*.
4. Click *OK*.

Now the plot shows many contours around the inlet illustrating the higher speed flows at ebb tide through the inlet.

### 5.6.2 Color Filled Contours

---

Another useful contour option is the use of filled colors. To see color filled contours:

1. Click the *Contour Options*  macro.
2. In the lower right section, set the *Number of intervals* to 25.
3. In the upper left section, turn on the *Color fill between contours* option.
4. Click the *Color Options* button. Change the *Color Method* to *Intensity ramp*. Click inside the *Default color* box and choose a ramp color.
5. Click *Reverse* so that higher intensity will be related to higher speeds.
6. Click *OK* to close the color options.
7. Click *OK* to close the *Contour Options* dialog.

The display will refresh, and should look something like Figure 5-6.

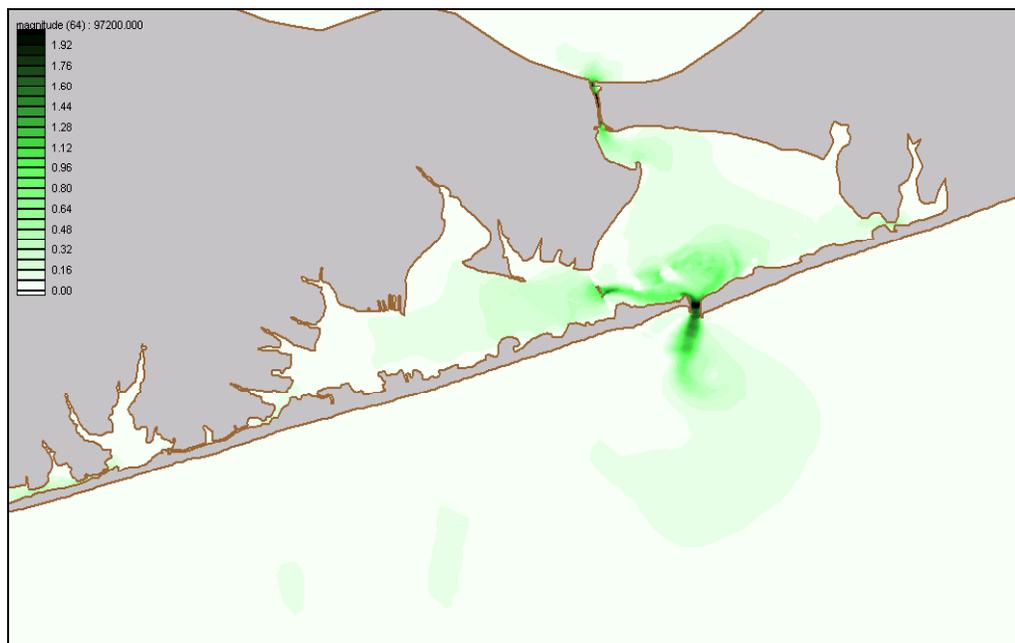


Figure 5-6. Color filled contours of velocity magnitude.

### 5.6.3 User-Defined Palettes

The two most common options for color contours are the *Hue Ramp* and the *Intensity Ramp*. The hue ramp allows a range of different colors, while the intensity ramp (as seen in the previous example) varies the shade of a single color. SMS also supports more color flexibility in the form of user-defined color palettes. The palette option allows you to specify any number of colors in any order. To illustrate the use of palettes:

1. Click the *Contour Options*  macro again and select *Color Options* button.
2. Change the *Contour Method* to *User defined*.
3. Click the *New Palette* button. A number of pre-defined values are available. Choose *Ocean* as the *Initial Color Ramp* and click *OK*.
4. Inside the dialog, choose the *Create Color*  tool and click in the middle of the color palette range.
5. Choose the *Select Color*  tool and double click on the new color bar. (Note that this tool also allows you to drag the colors around.)
6. Set the color to red and click the *OK*.
7. Click the *OK* button to close the *Color Options* and *Display Options* dialogs.

The display should now look similar to Figure 5-7. While this ramp probably would never be used for a report, this procedure illustrates the process of creating and using user-defined palettes. If you create a palette that you like, you can save it from within the *Color Options* dialog. You can then open it at a later time in a new project.

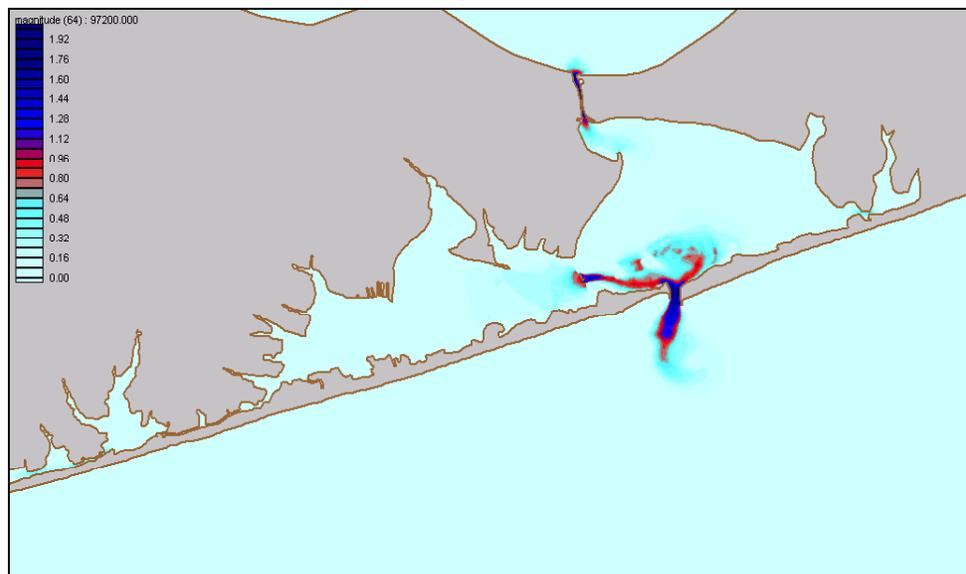


Figure 5-7. Contour plot using a user-defined palette.

## 5.7 Creating DXF Contour Lines

---

*SMS* can save mesh contour lines to a *DXF* file. This can be useful when you want to use the *SMS* contours with other *DXF* data in *AutoCAD* Or another package that supports the *DXF* format. To save your contour lines as a *DXF* file, first you must convert them to *DXF* entities. To do this:

1. Click the *Contour Options*  macro, reset the *Contour Method* to *Normal linear contours*. You must be using linear contours to export them to *DXF*.
2. For clarity, change the color ramp to a Hue ramp, set the number of contours to 10 and click *OK*.
3. Switch to the *Map*  module.
4. Select *DXF | Data -> DXF* command.

The contour lines have now been converted to *DXF* lines. To see the *DXF* data, and save it to a file:

1. Click the *Display Options*  macro, turn off the *Contours* option in the 2D Mesh tab, and click *OK*. The color contours will disappear, but the *DXF* contours will still be visible.
2. Select *File | Save as...* and set the *Save as type* to “AutoCAD Files (\*.dxf)”.
3. Enter the name “Shin\_dxf” and click the Save button.

The contour lines have been saved as an *AutoCAD DXF* file. You can now open this file into any application that supports *DXF*. Other entities that get saved to the *DXF* file (when they are being displayed inside *SMS*) include mesh nodes and mesh elements. The *DXF* layers can be seen in the *DXF* tab of the Display Options dialog. To delete the *DXF* data:

- Choose *DXF | Delete* (and click *Yes* to confirm). Then return to the Mesh  module and turn your contours back on.

## 5.8 The Data Calculator

---

*SMS* has a powerful tool, called the *Data Calculator* that creates new scalar data sets by performing operations on existing scalar data sets. The initial data for *ADCIRC* included bathymetry from mean sea level. We want to create a dataset of the actual depth at each node.

The actual depth is the bathymetry depth plus the water surface elevation.

To create the *Actual\_Depth* data set:

1. Select *Data | Data Calculator*. You will see a list of all current scalar data sets. In addition, *SMS* provides a couple of utility data sets that hold the x- and y- coordinates for the nodes.
2. Highlight the *elevation* data set and click the *Add to Expression* button. The *Expression* will show “a”, which corresponds to the *elevation* or depth below mean tide level.
3. Click the *add “+”* button.
4. The data calculator can perform its operation on a single time step of transient data, or all timesteps. We want to find the actual depth for all timesteps, so Highlight the *water surface elevation(63)* data set and select the *Use all timesteps* toggle box and click the *Add to Expression* button. The *Expression* will show “a+b:all”. The “b:all” corresponds to the water surface values.
5. In the *Result* field, enter the name *Actual\_Depth* and then click the *Compute* button. *SMS* will take a few moments to perform the computations. When it is done, the *Actual\_Depth* data set will appear in the *Data Sets* window.
6. Click the *Done* button to exit the *Data Calculator* dialog.

The *Actual\_Depth* data set can be seen as part of the *Generic Datasets* solution. (Note: you will probably need to change the contour options to view this data set since you have been viewing velocity magnitudes with a set range of 0-2. In the area of the inlet a good range for the actual depth is 0-25 m.) The newly created data set has all the time steps the water surface elevation data set had and can be treated just as any other scalar data set.

## 5.9 Vector Plots

*SMS* can be used to create plots of vector data as well as scalar data. Vector plots are useful for looking at velocity directions and flow patterns. The default option is to display all vectors at a constant length. To turn on the vectors:

1. Click the *Display Options*  macro.
2. Turn off the *Contours* option and turn on the *Velocity vectors* option. These options can be on together, but we’ll turn off contours for now to decrease screen clutter.
3. Click the *OK* button to close the *Mesh Display Options* dialog.
4. Select the *Shinnecock* solution (that is where the current vector data set is).

The screen will refresh, displaying one vector on each mesh node. As seen in Figure 5-8, the image can become cluttered when using the default vector options

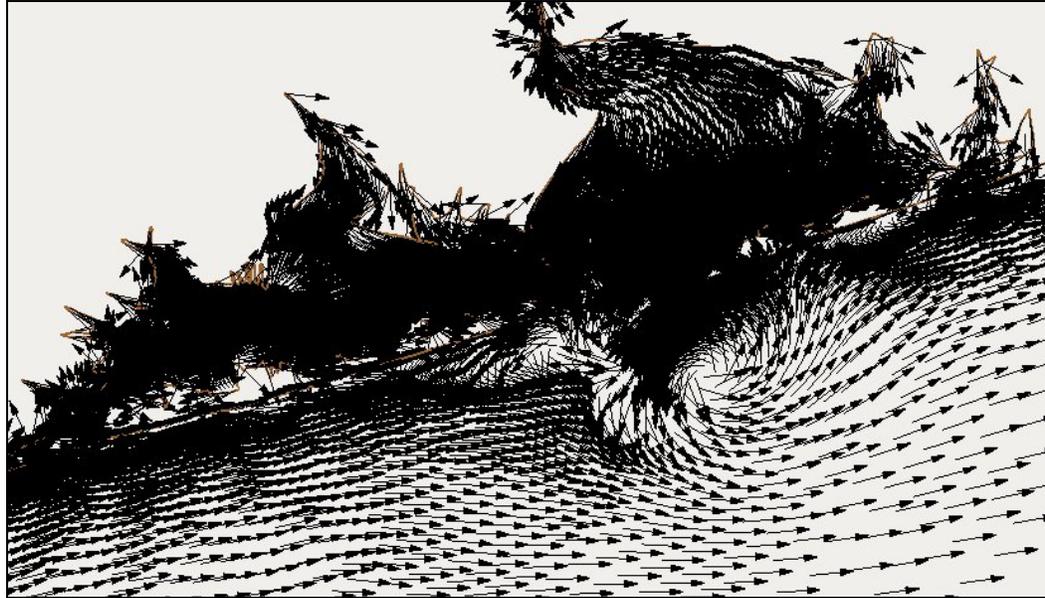


Figure 5-8. Velocity vectors turned on.

### 5.9.1 Vectors on a Grid

---

To reduce the clutter on the screen and make the image more informative, *SMS* has an option to specify how densely the vectors should be displayed. You can set *SMS* to display vectors on a grid. The grid is defined in pixels, or screen points. In this way, when you zoom in, you get more vectors per unit area of the model and when you zoom out, you get fewer. To set up vectors on a grid:

1. Click the *Vector Options*  macro.
2. In the *Arrow Placement* section, choose the “Display vectors on a grid” option. Set the x and y pix values to 15. Then click *OK*.

*SMS* will display vectors at 15-pixel (screen point) spacing. Figure 5-9 shows how this option looks at two different scales around the inlet.

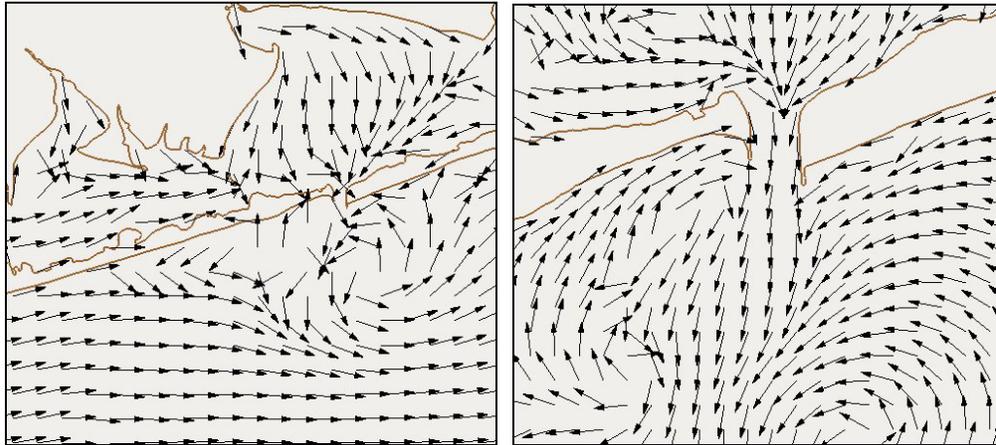


Figure 5-9. Vectors on a grid.

### 5.9.2 Variable-Sized Vectors

All of the vectors displayed in the plots above used constant length vectors. Each vector was drawn to be 35 pixels long. You often want to see relative differences in vector magnitude in addition to vector direction. There are two available options for setting variable-sized vectors. The vectors can be scaled based the vector magnitude, or you can specify a minimum and maximum display length. To set up a vector size range:

1. Click the *Vector Options*  macro.
2. Under the *Shaft Length* section, select the *Define min and max length* option and set the range to 10 and 45. All vectors will be scaled between ten and forty-five pixels. The minimum size of 10 ensures that all vectors will be long enough to show a clear direction. The maximum size of 45 ensures that no vectors will overpower the image.
3. Click the OK button to close the *Vector Options* dialog.

The display will refresh, showing vectors of varying size, as shown in Figure 5-10.

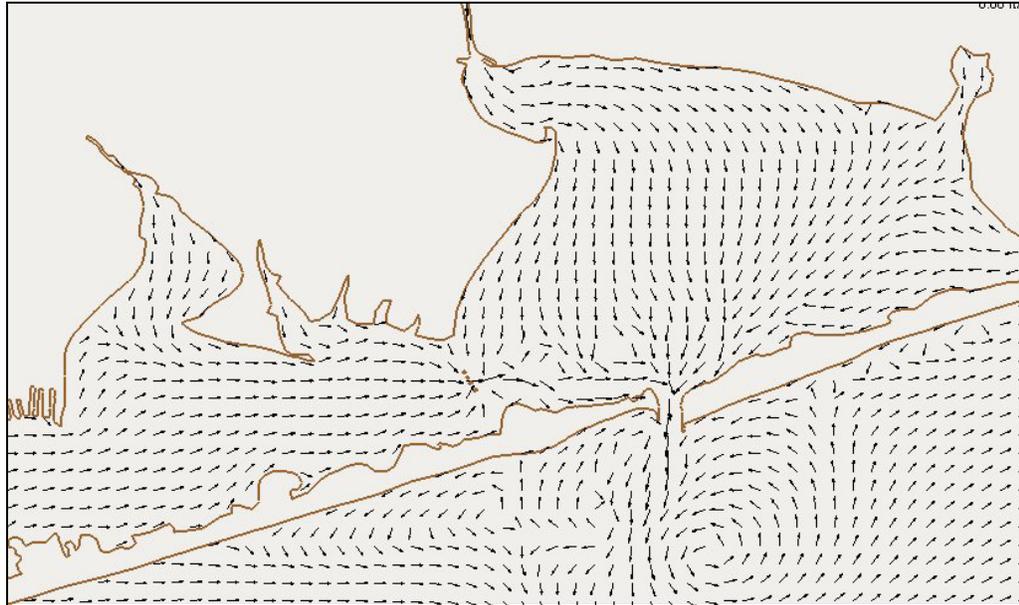


Figure 5-10. Vectors of varying size.

You may want to experiment with scaling vectors based on their magnitude. This option gives a clearer representation of the relative magnitudes, but large velocity vectors (such as through the inlet), may become very large and clutter up that portion of the image.

### 5.9.3 Vectors in Color

---

The last vector option that will be discussed is to change the color. Usually, black vectors are sufficient for display purposes, especially when the contours are also turned on. There are times, however, when you want to see the vectors in color. Vector colors are set up just like contour colors. To set up vector colors:

- In the *Vector Options* tab of the *Display Options* dialog, click the *Setup Colors* button. Set the colors any way you see fit and click *OK*. Click *OK* again to close the *Vector Options* dialog.
- Turn off vectors before going to the next section.

## 5.10 Observation Coverage

---

This section describes the observation coverage type. Using this coverage *observation profile* plots are created to look at scalar data along an arc. To create an observation arc:

1. Switch to the Map  module.

2. Choose *Feature Objects | Coverages* and change the *Coverage type* to *Observation*. Then click *OK*.
3. Choose the *Create Feature Arc*  tool from the *Toolbox*.
4. Zoom into the inlet and create an arc across, starting just to the left on the barrier island and ending just to the right on the barrier island, as shown in Figure 5-11.

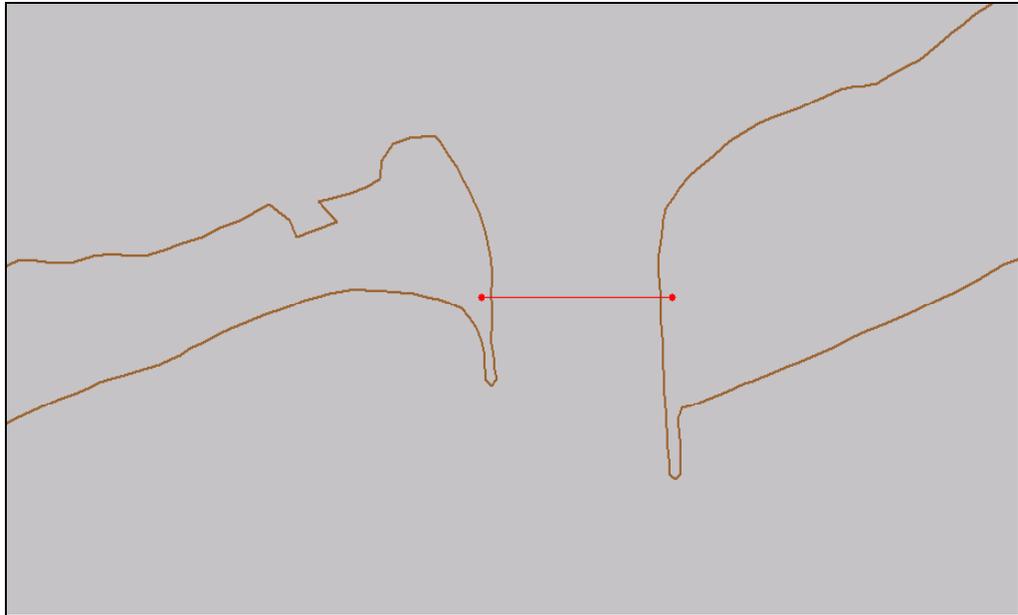


Figure 5-11. The location of the feature arc.

When observation plots are drawn, they will use the name and color associated with the observation arc. To change the name and color of the arc:

1. Choose the *Select Feature Arcs*  tool from the *Toolbox*.
2. Double-click on the profile arc.
3. In the *Observation Arc Attributes* dialog, change the name of the arc to *Inlet Cross Section* and change the color to whatever you want.
4. Uncheck the toggle box in the *Obs* column of the spread sheet.
5. Click the *OK* button to close the *Observation Arc Attributes* dialog.

### 5.10.1 Creating a Profile Plot

You will create a profile plot to see the variation in velocity magnitude across the inlet. To create the plot:

1. Select the plot wizard tool .
2. Select the *Observation Profile* option and click *Next*.
3. Select the *Use selected data sets* option and then select “magnitude (64)” in the lower left corner of the dialog. On the right side, leave the profile arc you just created checked and click the *Finish* button. The plot should now appear on your screen.
4. Right click on the plot and you will see a menu of options for adjusting the plot. The first item allows you to change the data plotted. The other options allow you to change axes, labels, etc. Play with these options until you are comfortable.
5. Click on the x at the upper right corner of the plot to close it when you are done.

### 5.11 Flow Trace Animations

---

The last post-processing technique that will be explored in this workshop is the creation of flow trace animations. These animations give an understanding of flow patterns by randomly placing virtual particles throughout the mesh and following them through it. To create a flow trace animation of the area near the notch:

1. Be sure you are in the *Mesh*  module.
2. Grab the lower right corner of the graphics window and make it about one half of its size.
3. Zoom in on the inlet and bay as you did earlier (Figure 5-5).
4. Select *Data | Film Loop*.
5. Select the filmloop type *Flow Trace*. Click the *Next* button.
6. In *Run Simulation from Time* choose 86,400 and run to 172,800.
7. Change the *Number of frames* to 48.
8. Click *Next*.
9. Change the *particles per object* to 1 and the *decay ratio* to 0.1.
10. Click *Next* and then *Finish*. You will see the images flash on the screen as the animation is generated.

11. When the animation has been fully generated, SMS will launch it in another application. If it does not launch, you don't have the application "pavia.exe". Download it from the SMS web page or play the avi file in another media player.



Figure 5-12. Flow trace animation.

The animation speed can be changed to give you the desired result. You can also play the animation in sequence or forward and backward. You should experiment with these options, and create multiple flow trace animations using different values for the options in the bottom right section of the *Film Loop Options* dialog. When you are done with this section:

- Click the *Done* button to close the *Film Loop* dialog.

## 5.12 Conclusion

You are now familiar with many of the post processing options that are available for a finite element solution. You may continue to experiment with SMS or you may exit the program. To quit *SMS* at this point:

1. Choose *File | Exit*. If prompted to confirm, click the *Yes* button.