Viewing and Post-Processing Results

Calibration of numerical models requires use of post-processed model output in order to compare with measurements and make incremental changes. This section gives guidance on how to extract and process data from model results, and examples of measured data to compare results with. Alternatively, the model results can be exported, and comparisons can be made using different programs, such as Matlab, Tecplot, but some options within SMS can help with repetitive testing and comparisons.

Water Levels & Currents

The first stage in calibration of a numerical hydrodynamic model is to accurately represent measured tides (or water levels) and currents. The number of calibration measurement sites required for sufficient model calibration increases with larger domains and increased complexity in processes.

Of the three main data sources required for hydrodynamic calibration (geomorphic, forcing, and field data), inaccurate knowledge of, or mistakes in conversion between datums is the most common underlying factor in poor model calibration. Model calibration cannot be achieved without proper representation of datums. Some examples of field data typically used for calibration of hydrodynamics include fixed water level gauges, fixed Acoustic Doppler Velocimeters (ADV) and Acoustic Doppler Current Profilers (ADCP) and roaming or boat-mounted ADCPs. Fixed data can be assessed over a time series, whereas boat-mounted data are collected at specified locations over time intervals. If roaming data are collected continuously, they must be assessed spatially and temporally.

For hydrodynamic data, the most common forms of measurements are recorded either temporally at a station, temporally at a fixed spatial extent, or spatially at relatively singular times. For example, a tide gauge is stationary, but a Horizontal Acoustic Doppler current meter (H-ADCP) may collect temporal measurements at multiple points. Because the CMS is a 2D model, ADCP measurements at multiple locations should be depth-averaged, as would be an upward-looking ADCP, for model and data comparisons. Similarly, a field campaign of depth-averaged current measurements collected at transects are all contemporaneous, but there are multiple time periods to compare.

With forcing data and corresponding datums, the accuracy and detail of the geomorphic parameters, such as bathymetry, sediment characterization, and structure definition, also need to be checked before hydrodynamic calibration can be achieved. The most important of these is bathymetry, which can have several issues typically associated with quality control, data density, interpolation, and grid resolution and domain size. A good example of this is a poorly resolved small channel in the far reaches of a bay, such as that in the northern portion of the Shark River estuary (Figure 1). Although lower velocities may be passing through this channel, and there may be little interest in morphology change here, it may be integral to providing the conduit of tidal flow to a shallow bay platform.

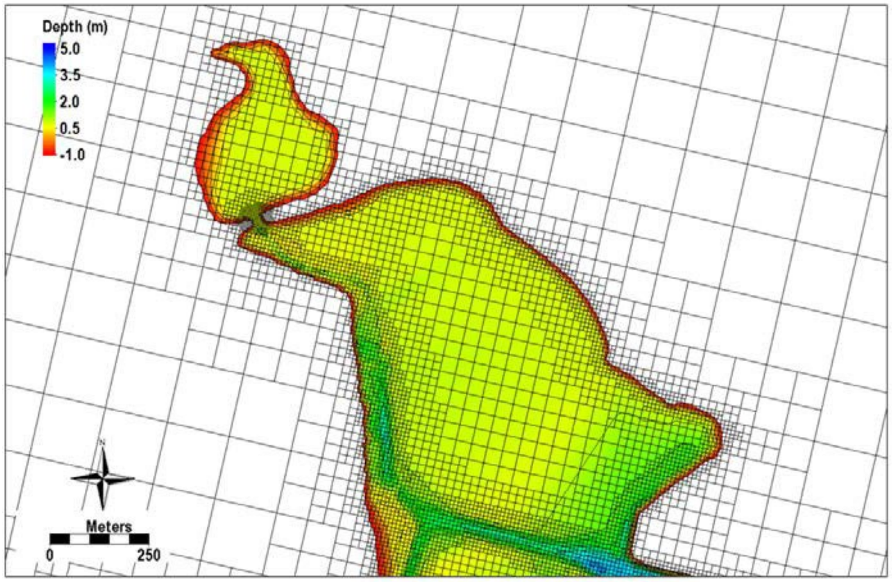


Figure 1. Northern portion of the Shark River estuary.

Figure 2 shows the closest locations of water-level gauges to Shark River Inlet. Sandy Hook and Atlantic City, NJ, are two ocean pier-mounted gauges approximately 30 and 100 miles away from Shark River Inlet, respectively. Both NOAA gauges were evaluated against the Belmar tide gauge using the tidal constituents of one year, and Sandy Hook was found to have the closest amplitudes and phases. Measured tides from Sandy Hook were used to force the offshore boundary condition. The calibration period that the model was evaluated on was for the 13-hour tidal cycle over which field measurements were collected on 20 August 2009.

Figure 2. Location of NOAA (red) and USGS (blue) water level gauges.

Display Options

The results of the CMS calculated hydrodynamics can be viewed and compared to measured results in a number of ways. Scalar results are typically in planar view and are illustrated by color representing magnitude, and vector results by arrow lengths representing magnitude and direction. Color setting can be manipulated in the overall *Display Options*, or individually for each dataset.

- Open a solution file for the water level output (**_wse.h5*) and right click on the *Water_Elevation* dataset, select *Dataset Contour Options*.
- In the *Contours* tab, adjust the color settings and the value range to that in Figure 3.

Figure 4 illustrates the new color scheme for displaying the water surface elevation.

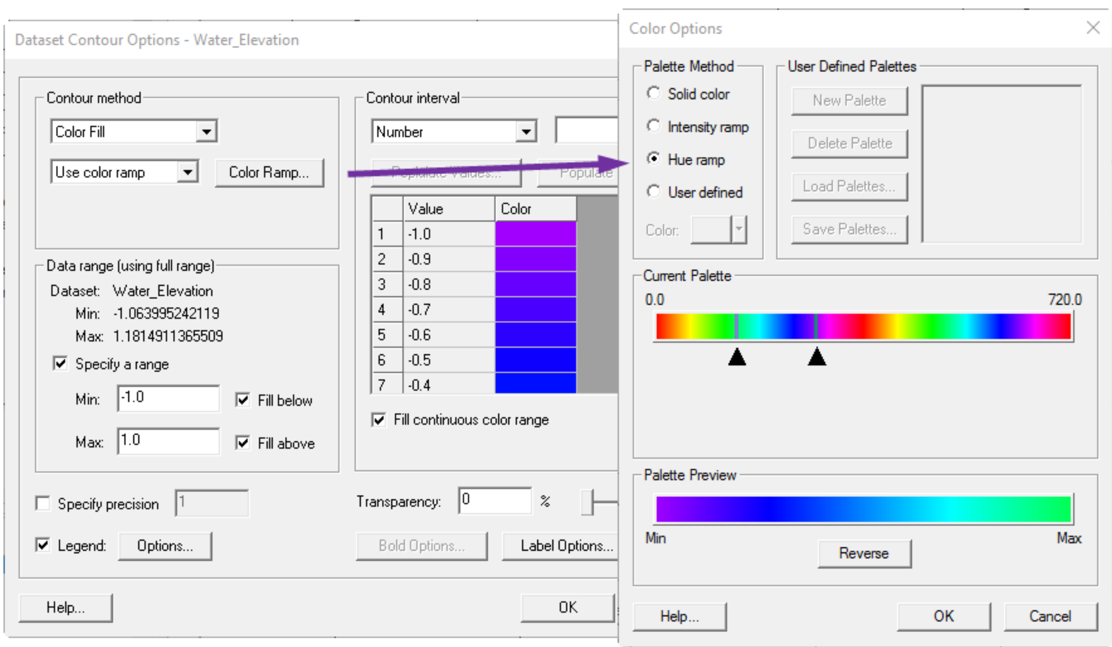


Figure 3. Contour tab settings for color display and value range. Note the blue colors were selected for the visible range.

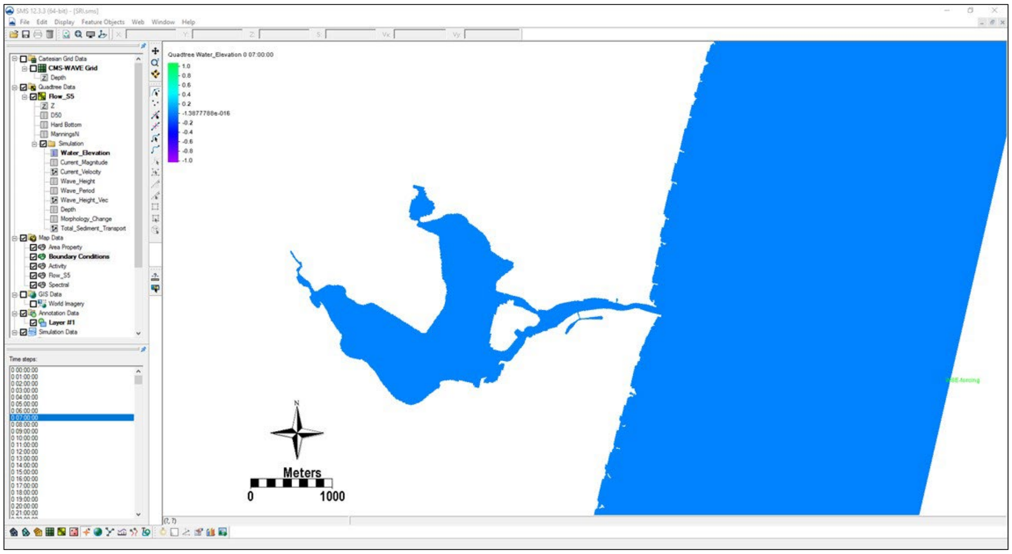


Figure 4. Water surface elevation color map modified.

Currents are automatically given in vector format as the solution files are generated. These can be converted into Vx and Vy components, or into magnitude and direction. If the model results are compared to simple magnitude measurements, then the magnitude dataset is appropriate. If the model results will be compared to measurements in E-W and N-S components, care should be taken to compare the correct derivative of the vector format. Because the vectors are based on the Cartesian grid's orientation, be sure to convert with respect to the axis difference.

- Right click on the current vector dataset and select Vector to Scalars. Select Vx and Vy, and this gives the x and y components of the current velocity. Current speed is saved in the Current_Magnitude dataset. The color component of the magnitude can be selected along with the vector of the current velocity.
- Open the *Display Options* and go to the *Quadtree/Vector* tab. Change the vectors to the below settings (Figure 5) to view the vectors over a prescribed gridding. (Note that individual vectors at each cell is difficult to visualize for the whole domain and only helpful when zoomed in very close.)

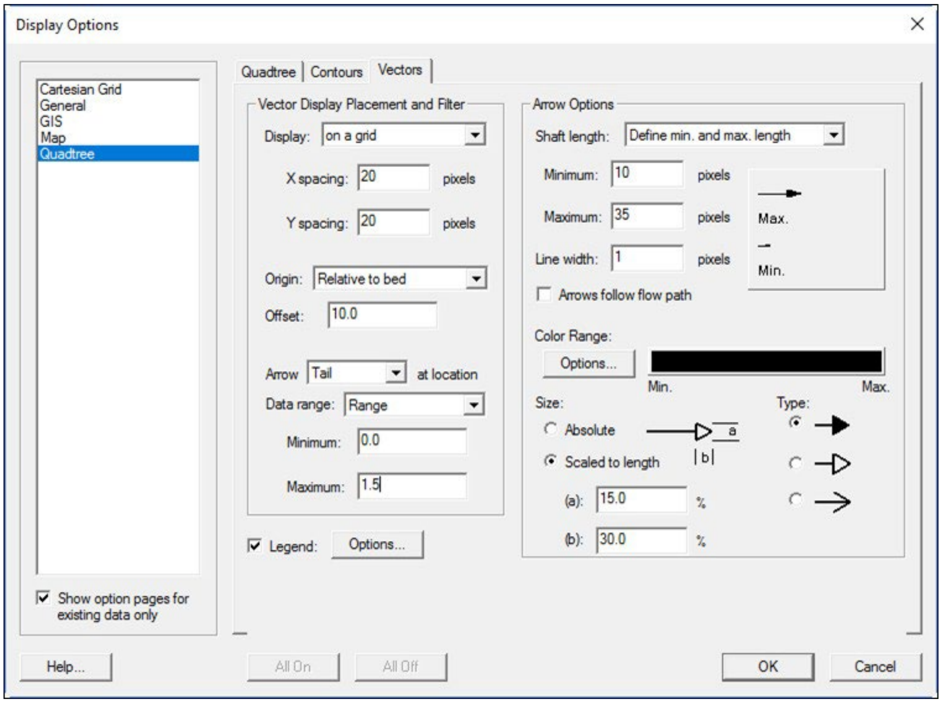


Figure 5. Vector Display options set for a gridded display across the domain.

Figure 6 shows an example of displaying current magnitudes with color contours and the current direction with vectors.

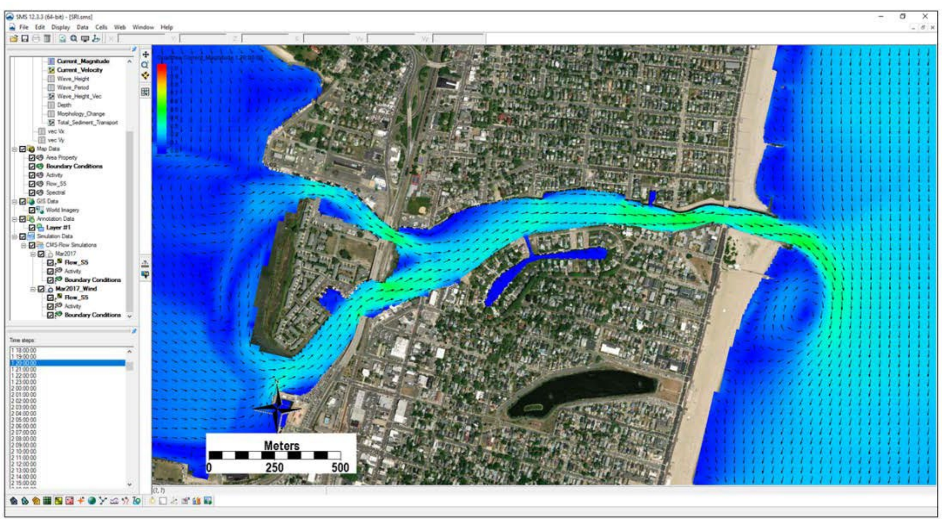
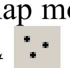


Figure 6. Vectors and color contours set for displaying current magnitude (color) and direction (vectors). Note that the vectors are slightly scaled on this setting to reflect magnitude.

Importing water level measurements into SMS

To compare the measured data within SMS, the measurements must be brought into the program and into a comparable format. As a simple example of comparing tidal data to calculated tides, the USGS Belmar tide data is given in the folder

Workshop\Final\Data in the Tide_Comparison_AUG09_Workshop.xls file. To import this data, it must be brought in as observational data within the *Observation Map Module*.

- In the Cartesian Module, go to *Display*, *Plot Wizard*, and select *Time Series Plot*. Select scalar, and the WSE as the dataset. (Note it takes some time to populate a plot in SMS.)
- Change the map module *Default Coverage* type to *Observation* and add a *Feature Point*  to the approximate location of the tide gauge shown in Figure 2.
- Right click on this feature point (Figure 7) and select attributes.

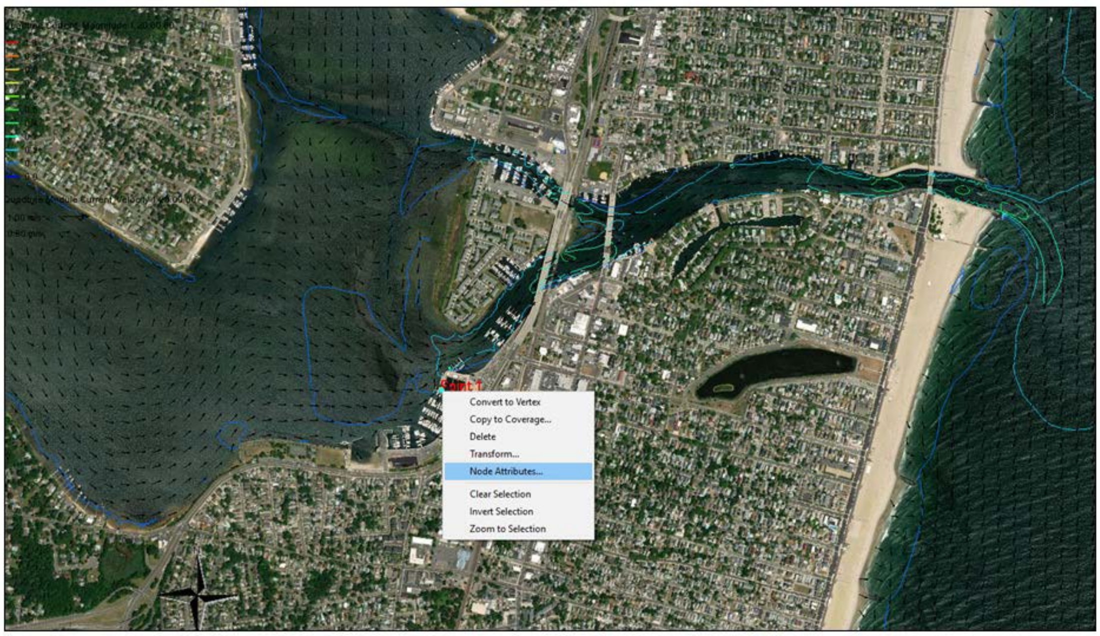


Figure 7. Observation point placed at the location of the Belmar tide station.

- In the *Attributes*, check the Trans box in measurements, and select 2D Cartesian under Module, and the Water_Elevation dataset (Figure 8). If your *Observe Point* is checked, an Options box should appear under the Time Series column. Select this and import the Belmar_Tide.xys file.

Time series of the measured water levels can be manually pasted here. A file of measured water levels is provided and available for *Import*: Belmar Tide Input.tsd.

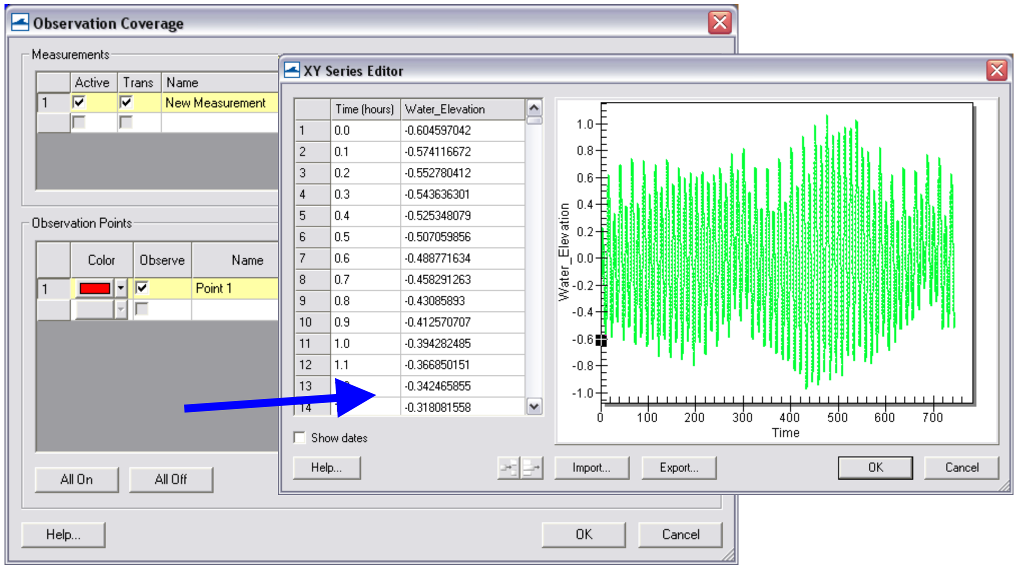
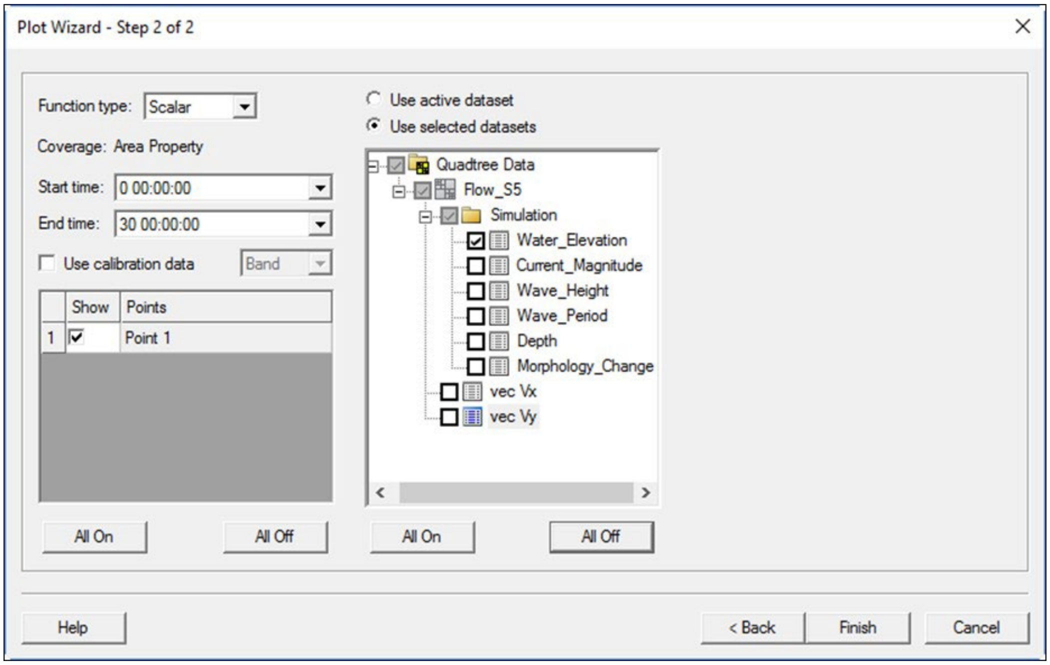


Figure 8. Input of time series water surface elevation data into an observation point.

Calculated water level plotting from Feature Points

Time series plots of water surface elevation can be created through the SMS plotting package. The steps to generate a plot are:

- Select *Display* | *Plot Wizard* | *Time* Series and select *Next*. The *Function* is *Scalar*, and the *Water_Elevation* scalar dataset should be selected with the full time period (Figure 9).



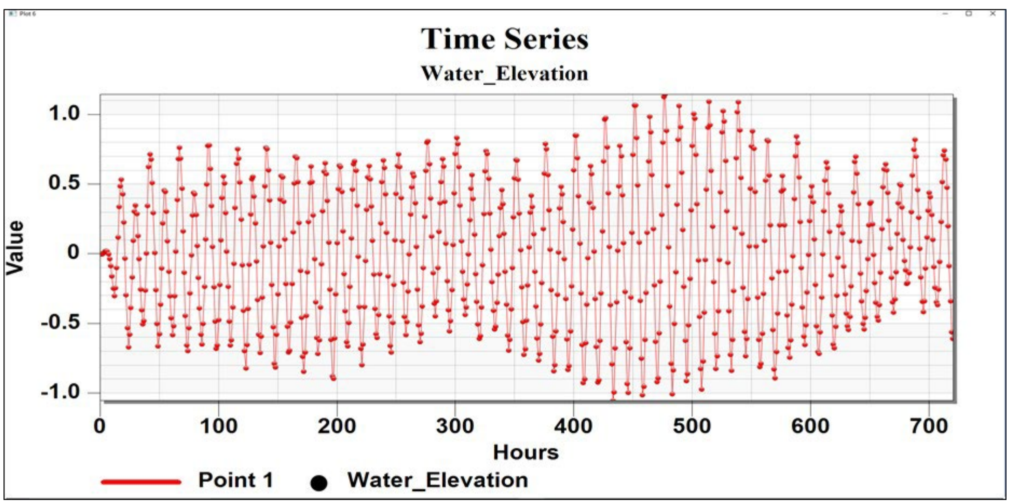


Figure 9. Plot Wizard for time series data (top); Time Series plot of Water Elevation (bottom).

- The plots can be updated by right clicking the graph window and selecting *Plot Data*. Also, the time frame can be modified.
- Changing the location of the *Feature Point* by dragging the point will also automatically update any graph open within SMS. Note that moving *Feature Points* that are extracting large datasets may take a considerable amount of time to reload.

Extensive statistical analysis of time series water levels is not available in SMS. However, it is convenient to extract this information from SMS which can be used in other software or Microsoft Excel.

- To extract the raw data for use in other software right-click on the graph window and select *Export*. In the *Exporting Time Series* window, select

Export|Text/Data, Export Destination|File (and specify a location) and select *Export*. Change the *Export Style* to *Table* and *Row vs. Column* to *Points/Subsets* (Figure 10). Export.

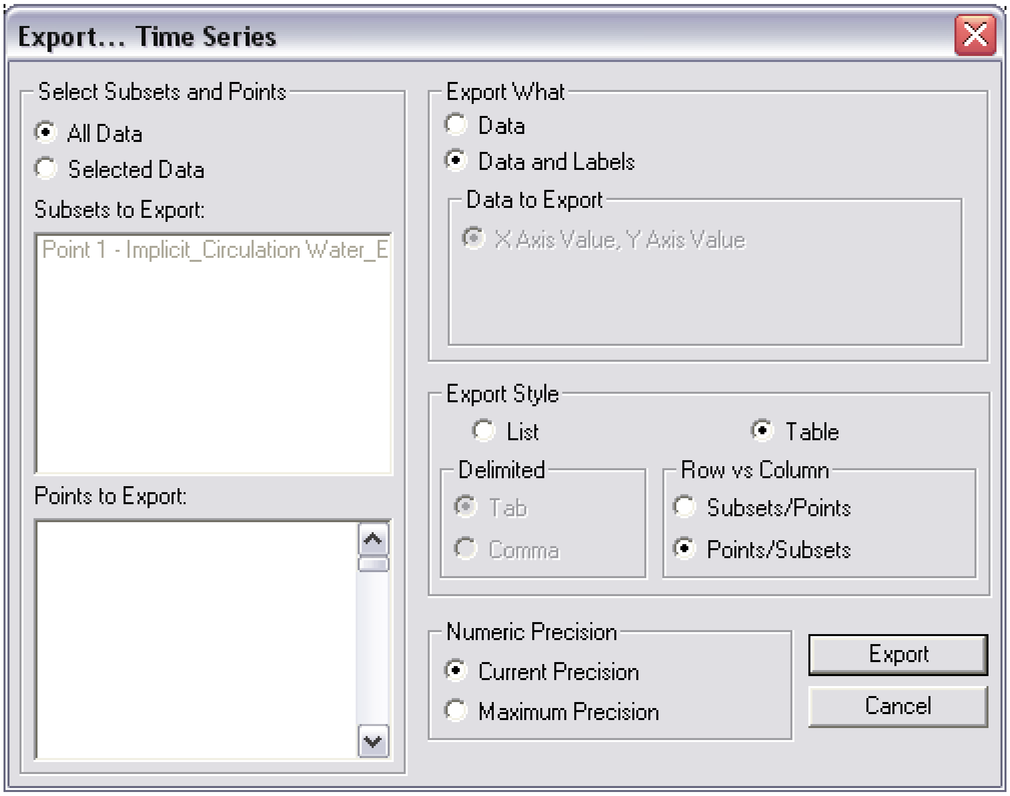


Figure 10. Export Wizard for plotted data.

- To view the raw data (and another way to extract) right-click on the graph window and select *View Values*. The data are presented in a table that can be highlighted and copied to another program.

Other options for the display plot can be found on the XMS Wiki (http://www.xmswiki.com/xms/SMS:2D_Plots).

CMS-calculated water level variation is compared with water levels from the Belmar gauge (location shown in Figure 2) and given in Figure 11.

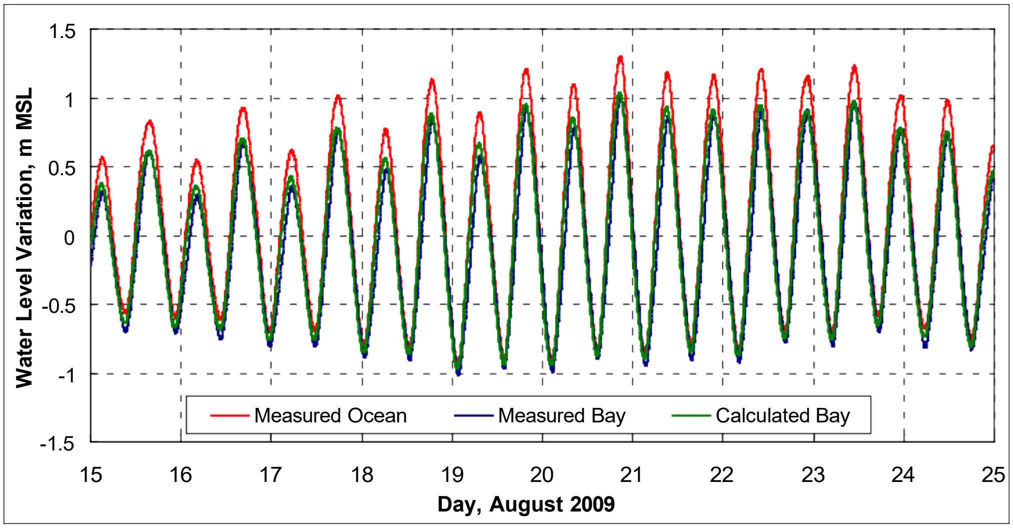


Figure 11. Observed time series of water level at Sandy Hook and Belmar ("Bay") and calculated water level at Belmar.

If time series data for the model were not set to the same interval as the measured data, there are filtering options within the data calculator. Reducing output (water level) from the solution:

- To create a 30-minute dataset of water levels from the original 6-minute output, go to *Data*, *Data Set Toolbox*, and under the *Tools* section, select *Temporal*, *Sample Time Steps* (Figure 12).

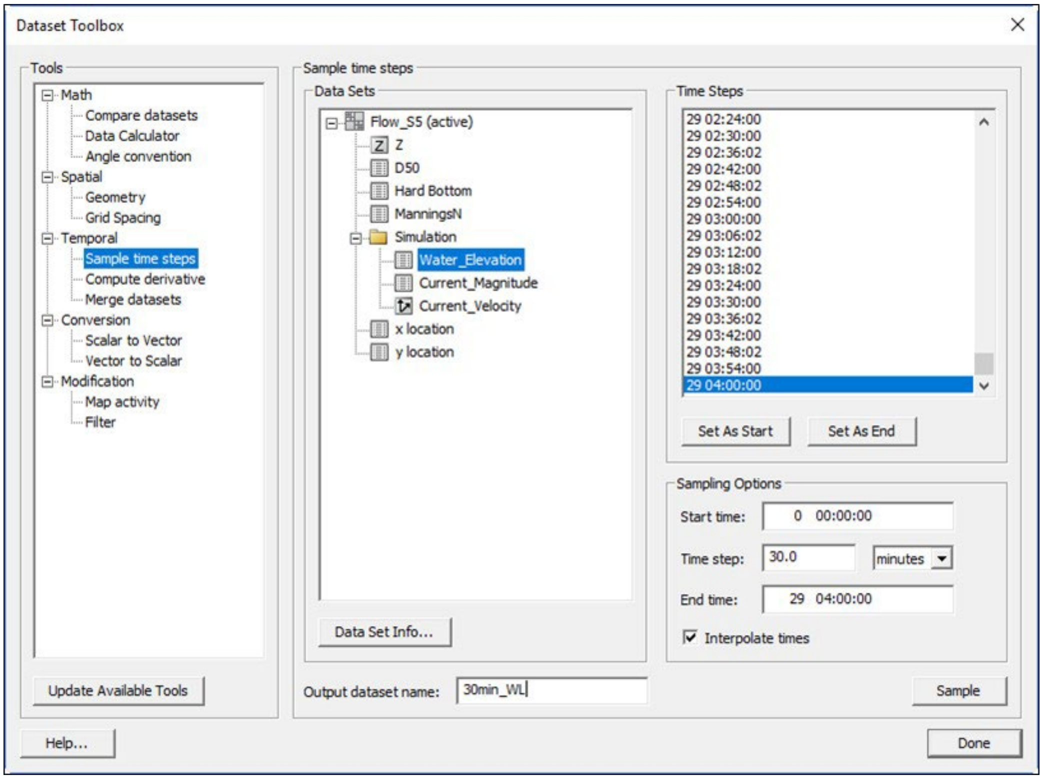


Figure 12. Sampling time steps of the Water_Elevation scalar dataset in the Dataset Toolbox.

- Select the *Water_Elevation* dataset under the solution data tree and create an *Output dataset name* at the bottom.
- Set the Start time to the beginning of the series and the end time to the end by selecting the time under *Time Steps* and selecting the associated button below. Change the time step to 30 minutes (or 0.5 hours), click *Sample*.

Depth-average down-looking ADCP current data were measured along three transects in Shark River Inlet during the calibration time period in August 2009. These data were processed with off-the-shelf software, where velocity was binned into depth measurements throughout the water column and converted into a .GIS file. To display preprocessed measured velocities, they must be converted into an XYZ format with additional data (velocity and direction, or *x*- and *y*-components, V_x and V_y) in additional columns. SMS will recognize both the XYZ data and assign extra scalar or vector datasets to the imported files.

Importing GIS-type files from D-ADCP current measurements output into SMS:

- Open the provided file, SRI082009_DAV_1300.GIS, in the folder *Workshop\Final\Data* and select *Use Import Wizard* (An excel version of the text file is also provided in the folder). Select *space/tab* delimited if not already checked and click *Next*, and select the X, Y, Z, Vector X and Vector Y columns (Figure 13). Change the Header of the Vector X & Y columns so they are the same (Vector), otherwise an error incurs when importing the file.

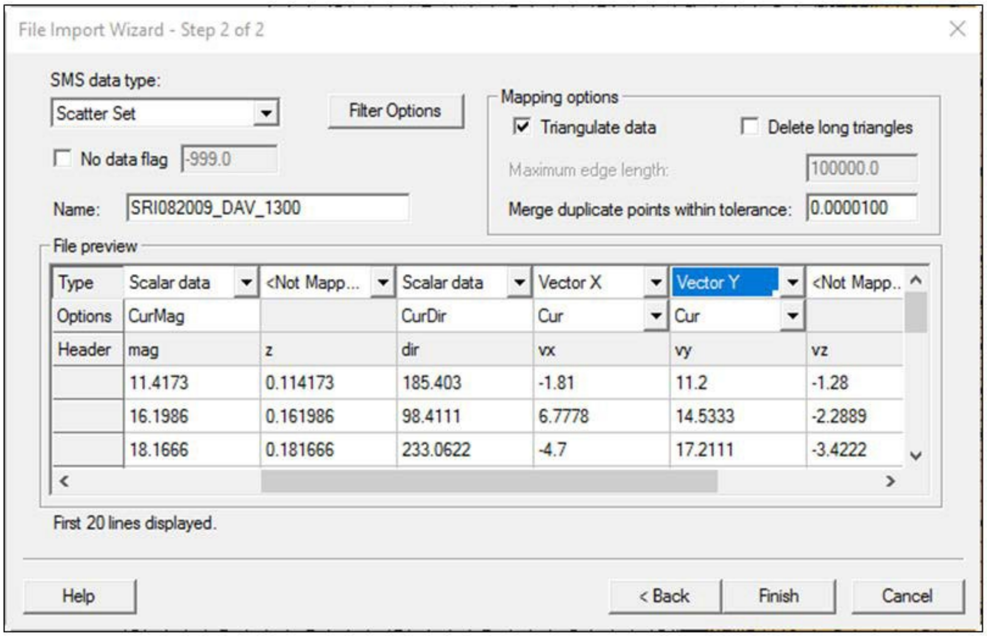


Figure 13. Read in vector files using Import Wizard.

All of the files were imported into SMS similarly as the time series data in the above section. All the data imported included X, Y, Z, Vx, Vy, Mag, and Dir for all three transects at a particular measurement time period. The files (under *Workshop\Final\Data*\SMS) have been converted into scatter datasets for quick import in SMS for viewing.

- Import (drag and drop) one or two of the .h5 files (e.g., *SHARK820_HOURLY -SRI082009_DAV_1100.h5*). In *Display Options*, under *Scatter*, select velocity vectors and adjust the vector settings tab (Figure 14).
- Calculated and measured vector components can be compared here. For a general estimate of comparison of the magnitude, convert one of the measurement files' vectors to magnitude and direction (right click, *vector to scalars*). This can be compared to the calculated magnitude for a rough estimate.
- The scalar value of the measured data is in centimeters. Create a new dataset in meters using the *Data Set Toolbox/Data Calculator* (Figure 15).

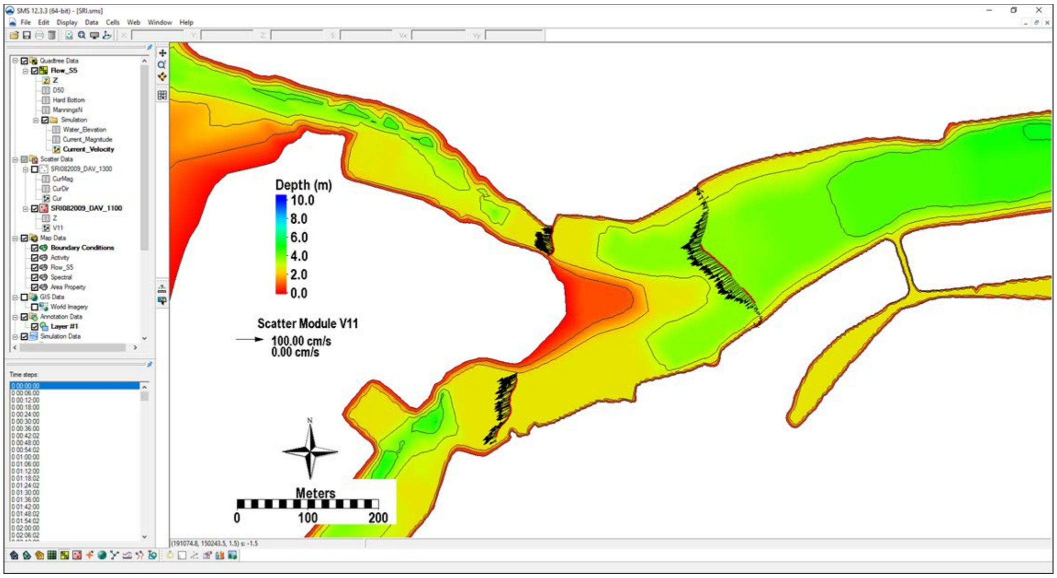


Figure 14. Example of the vector display along the measured transects. Note the vectors are scaled to the magnitude, and these scales can be manipulated in the Scatterset Vector tab.

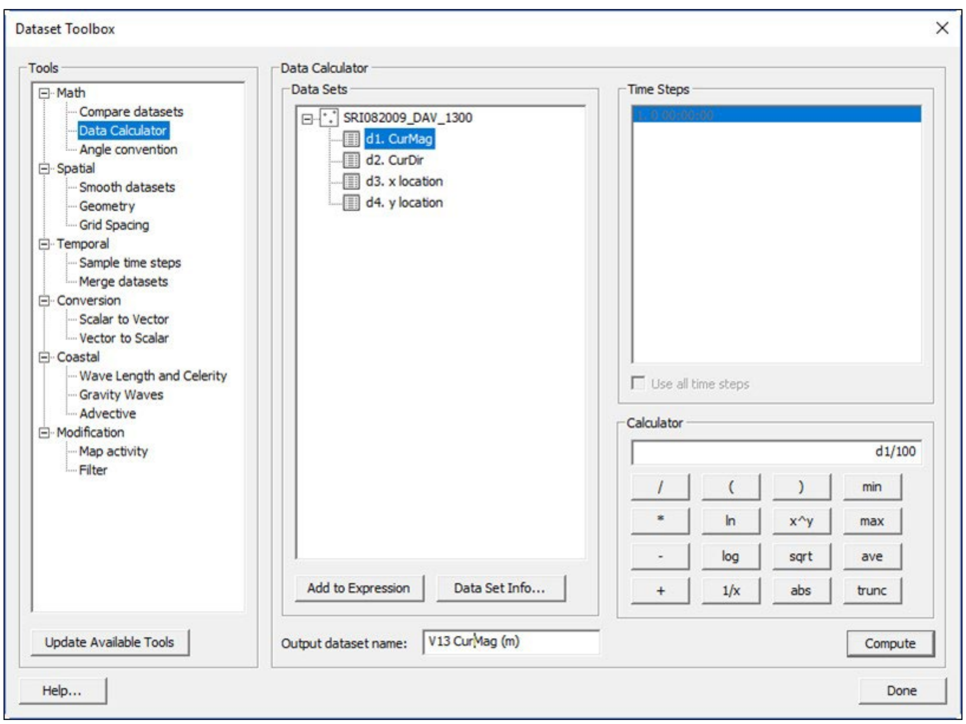


Figure 15. Data calculator used to convert the vector magnitude from centimeters to meters for comparison to the calculated velocity magnitude.

- To display the scatter points with colors representing the flow, go to the *Scatter* Tab under the *Display Options*, and select the *Use Color Contour Scheme* under *Points*. Make the points a size 10 and they will be visible and the color differences noticeable as in Figure 16.

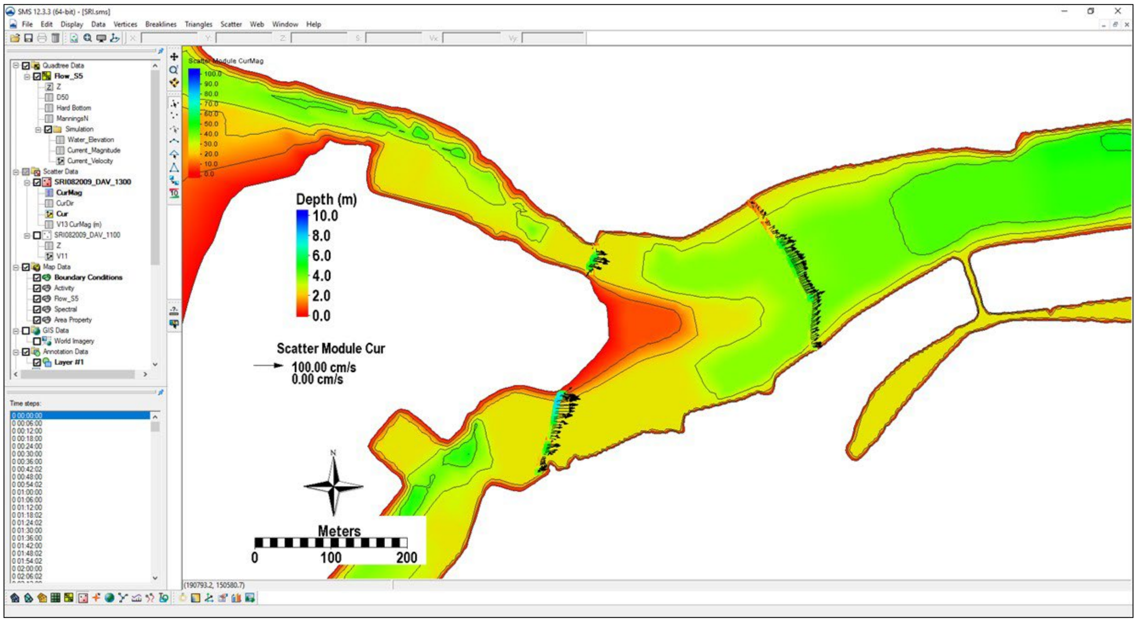


Figure 16. Color options turned on for scatter points.

To compare the measured current velocity data to calculated, the data must be in SMS in this format, where XYZ data overlay the proper region and the units of the measurements are the same. Both datasets can be viewed and extracted within the SMS through the use of observation arcs. The arcs must be set along the transect line and have enough points to illicit one value per cell, but not too many where there will be duplicate calculated values. The imported measured datasets are much denser than the cell coverage. Below describes how to plot a transect or observation arc, through the measured points that will display both measured point data and calculated data across a distance.

Displaying measured and calculated currents in SMS

- Following the above section, the current velocity file is displayed similar to Figure 17. Draw three observation arcs over each of the three transects. Directional arrows will display, and the direction of the arc should reflect ebbing or flooding. If the arrows face the flooding direction, flood currents will be positive, and ebb currents will be negative.
- Redistribute the vertices on the arcs (they should not have any yet) to a value similar to the number of cells across each transect. E.g., Transect 1 (main channel) should have 25-30 vertices.

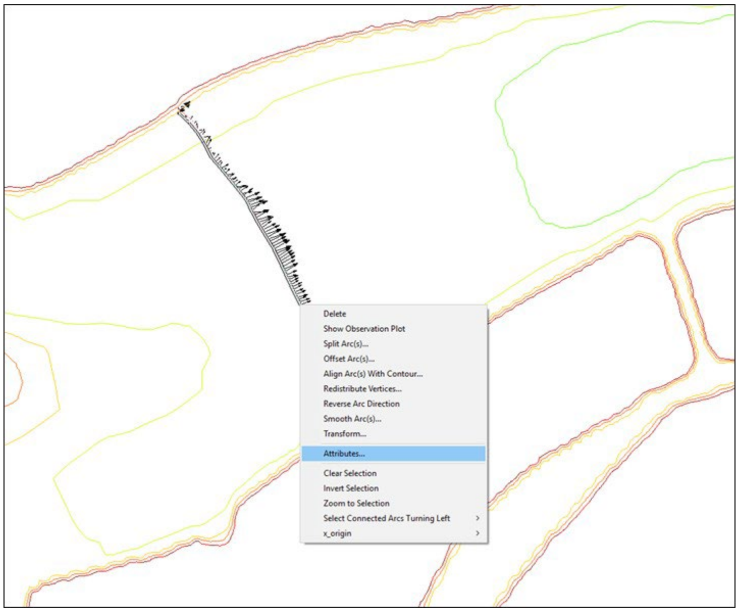


Figure 17. Example of a feature arc with arrows in the ebb direction. Redistribute Vertices, Reverse Arc, and other options are available in the drop-down menu.

- To plot, select *Plot Wizard* from *Display*, and select an *Observation Profile*. In Step 2, select one of the arcs (uncheck others), and under *Extra Profile* select *Model Intersections*. Figure 18 illustrates everything selected for plotting the measured and calculated data for the specific time of the transect measurement.

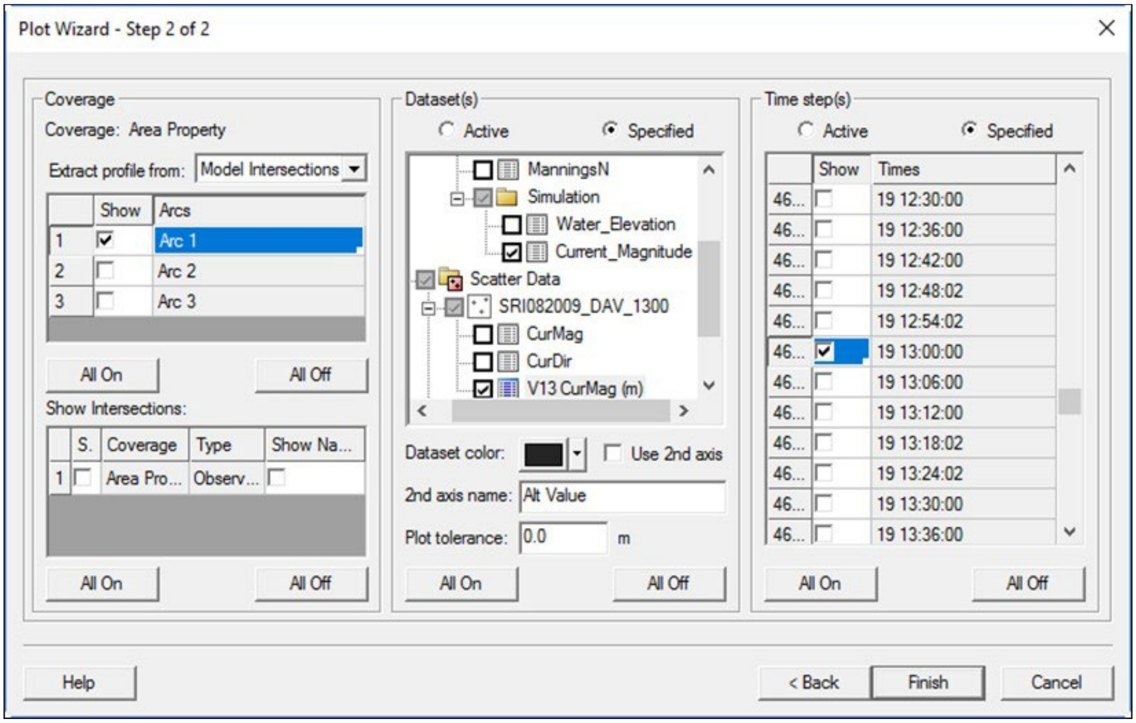


Figure 18. Plot Wizard data options for displaying measured and calculated data from an observation arc.

- Specify the datasets to be plotted, the *Current_Magnitude* under the grid and the *Vector Magnitude* under the scatterset data. Select the date and time the transect was measured, which was 20 August 2009 at 1PM GMT.
- The resultant plot is shown in Figure 19, where the closest points from the measured data were plotted as distance across the arc, and model grid cell centers are where numbers are extracted for the calculated data. Data can be extracted from the plot similarly to the ways defined in the water level extraction section at the beginning of this section.

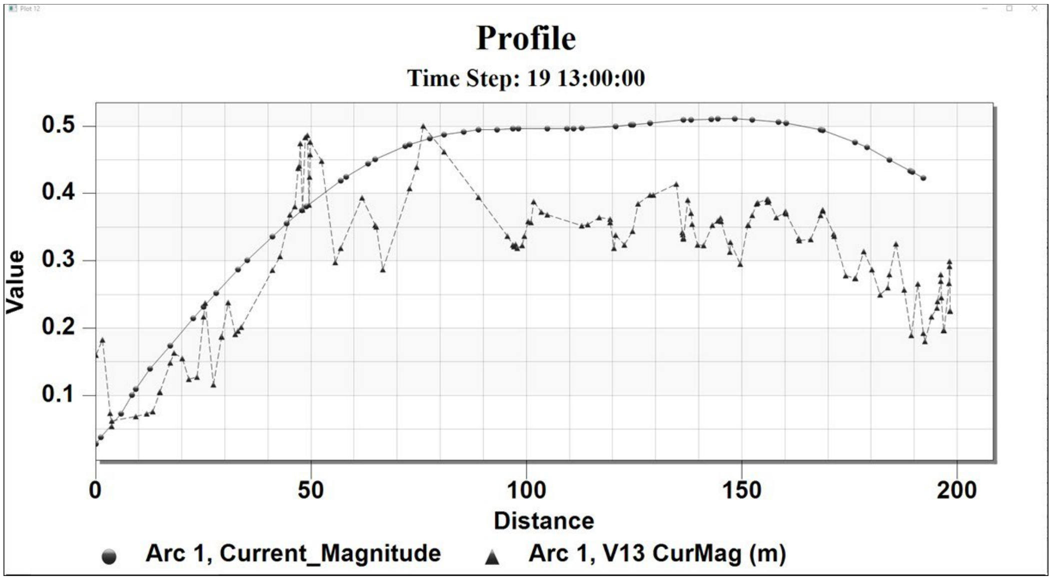


Figure 19. Plotted measured and calculated data from the larger observation arc to the East. Note that the positive direction reflects the observation arcs direction.

Displaying 2D current/sediment transport fields with wind/wave compass

Coastal circulation and sediment transport are primarily driven by wind and wave fields. When viewing/examining spatial and temporal variability of current and sediment transport, it will help the analysis to show wind and wave forcing simultaneously. In SMS a compass plot of wind or waves can be created with the display of current or sediment transport fields.

The following steps demonstrate how to plot a wind compass synchronized with temporally varying flow fields.

- Switch to the *Map Module*, right-click *Map Data*, select *New Coverage*, *Spatial Data*, and name the coverage, Wind_Compass (Figure 20).
- Create a feature point in the *Spatial Data* coverage.
- Click the *Select Feature Point* button and select the generated point, then right-click it and select *Add -> Compass Plot* (Figure 21).

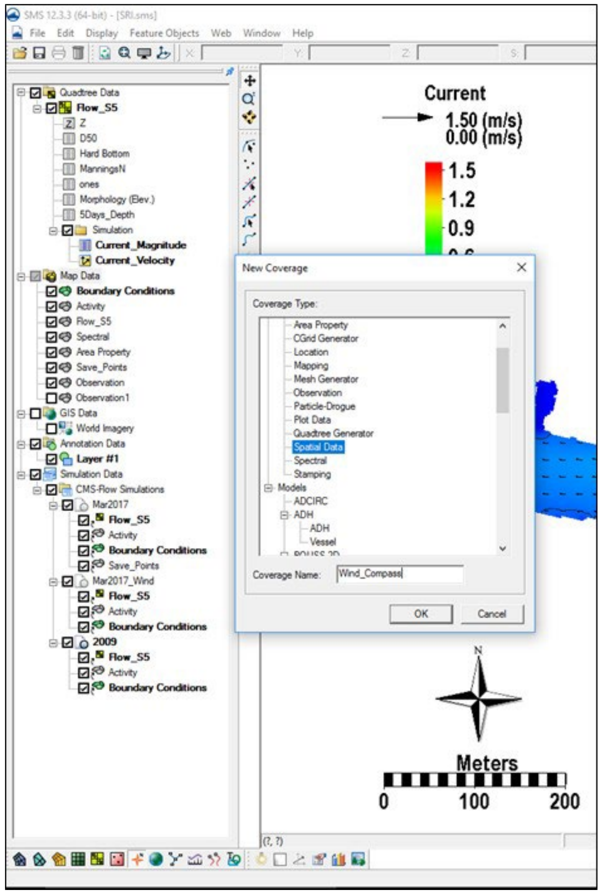


Figure 20. Generate a Spatial Data Covergae.

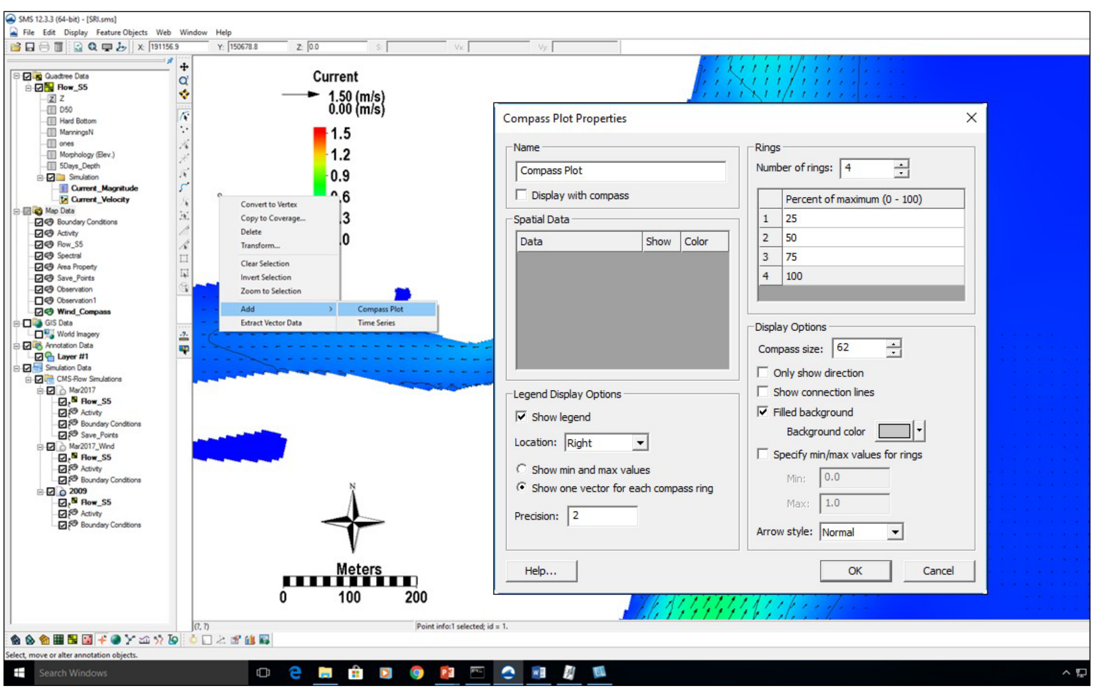


Figure 21. Add a Compass Plot.

- Click the *Select Feature Point* button and right-click it and select *Add -> Time Series*.
- Click *Import* to read "WIND" from a *tsd file or copy/paste wind information from an open Excel file (Figure 22).
- Click *Select Compass Plot -> Properties* and check the *Show* box under *Spatial Data* (Figure 23).

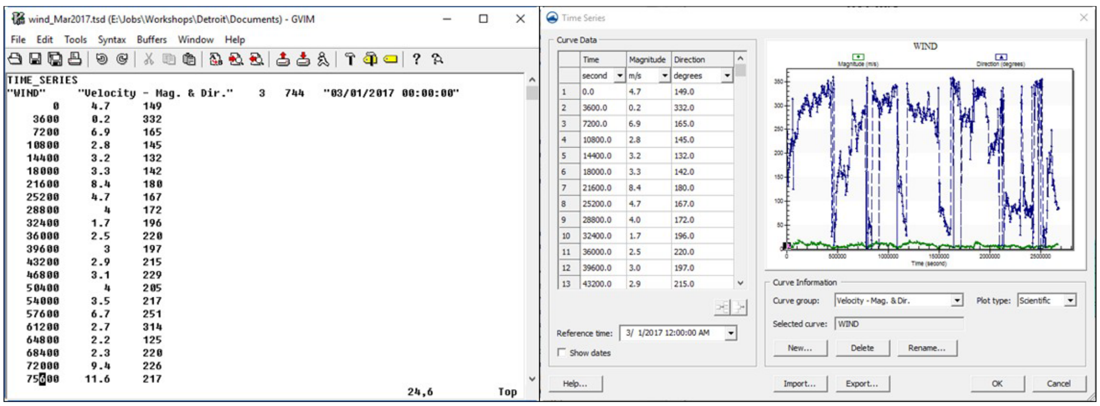


Figure 22. Read wind data.

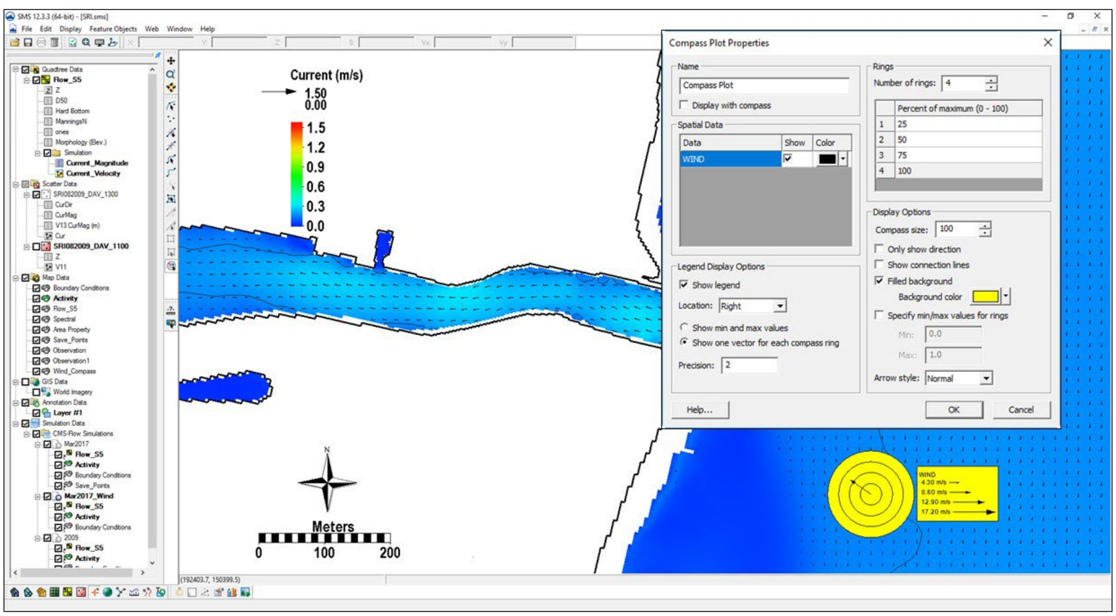


Figure 23. Link wind data with compass plot.

Sediment Transport

Sediment transport rates can be used to calculate sediment statistics, sediment pathways, fluxes across arcs, and balances over polygonal areas. Several sediment transport statistics can be calculated during the simulation and output at any time. These statistics include the net and gross total sediment transport rates. For additional details see the Simulation Statistics section.

Calculating Sediment Transport Roses at Observation (Save) Points

Sediment transport roses are plots in which transport is integrated over directional bins and plotted over a map of the site in order to observe the amount and direction of sediment transport at a specific point. A general picture of the sediment transport pathways can be obtained by plotting several sediment transport roses in the areas of interest. Below is a step-by-step example for Shark River inlet on how to plot sediment transport roses:

Setup Observational Cells

- Generate a *New Coverage* under *Map Module* and rename it as Save_Points.
- Right-click Save_Points and select *Type* -> *Model* -> *CMS-Flow* -> *Save Points* (see Figure 24).

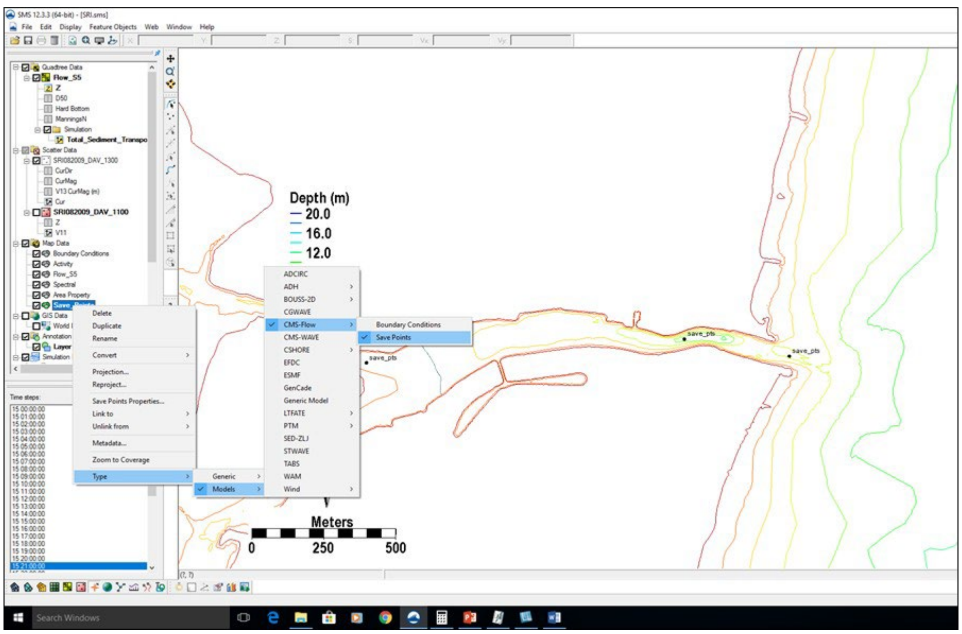
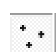


Figure 24. Generate a new map coverage for specifying observation cells.

- Select save points using *Create Feature Point*, .
- Drag Save_Points coverage under a CMS-Flow Simulation and save the project (Figure 25).

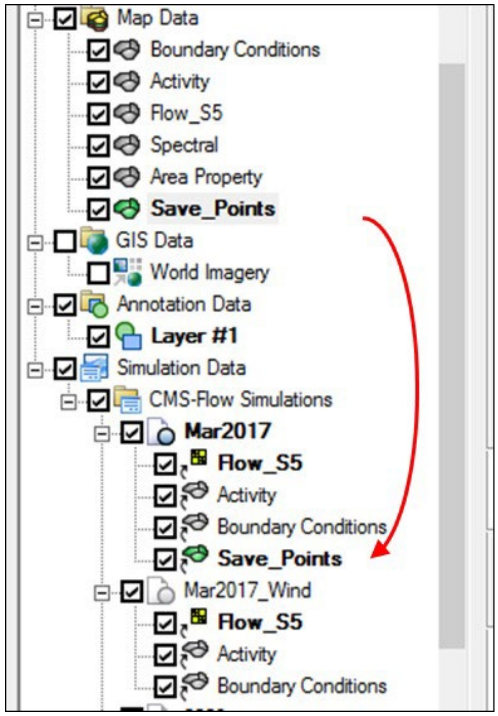


Figure 25. Drag Save_Points coverage under a CMS-Flow Simulation, Mar2017.

The CMS-Flow Control (Card) File will contain a section for the observational cells that look like this:

```
SAVE_POINT"save_pts" -1 191958 150381 HYDRO SEDIMENT SALINITY
*SAVE_POINT_OPTIONS_BEGIN
*SAVE_POINT_NAME                  "save_pts"
*SAVE_POINT_HYDRO_TOGGLE          1
*SAVE_POINT_SED_TOGGLE            1
*SAVE_POINT_SALINITY_TOGGLE       0
*SAVE_POINT_WAVES_TOGGLE          1
*SAVE_POINT_OPTIONS_END
```

Run the model

Once the model simulation has completed. The project folder will contain a sediment flux file with the extension *_trans.sp. This file contains the total sediment transport rates in the *x* and *y* directions in units of kg/m/s.

Plot the sediment transport roses in Matlab

Exporting a Georeferenced image in SMS

If the grid is telescoping, it may be difficult to plot the grid in Matlab. There are ways of plotting a telescoping grid in Matlab, but for the purposes of visualization, the easiest approach is to export a georeferenced image from SMS and plot the image in Matlab. This is done by

- Click on *File* | *Save as*… (see Figure 26)

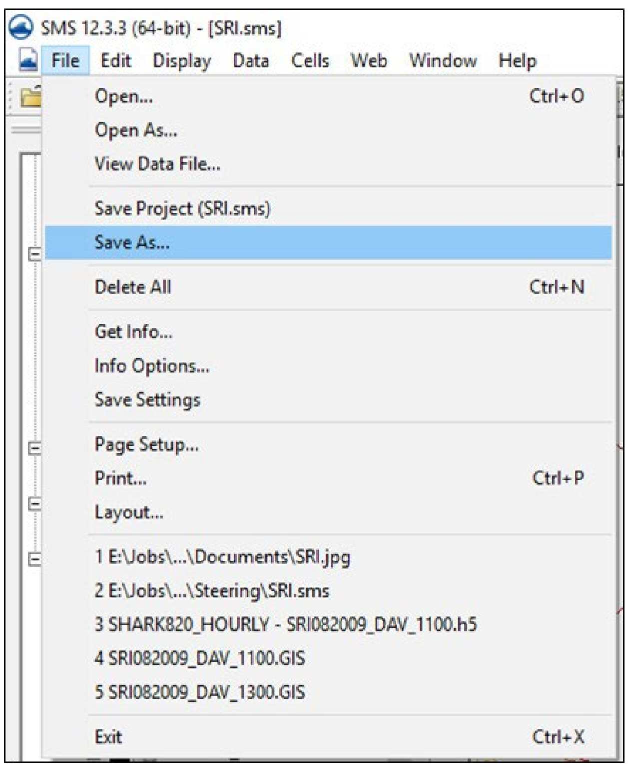


Figure 26. Opening the SMS Save As window.

1. Next to *Save As* type, select JPEG Image File. Navigate to the correct directory, enter a file name, and click *Save* (see Figure 27).

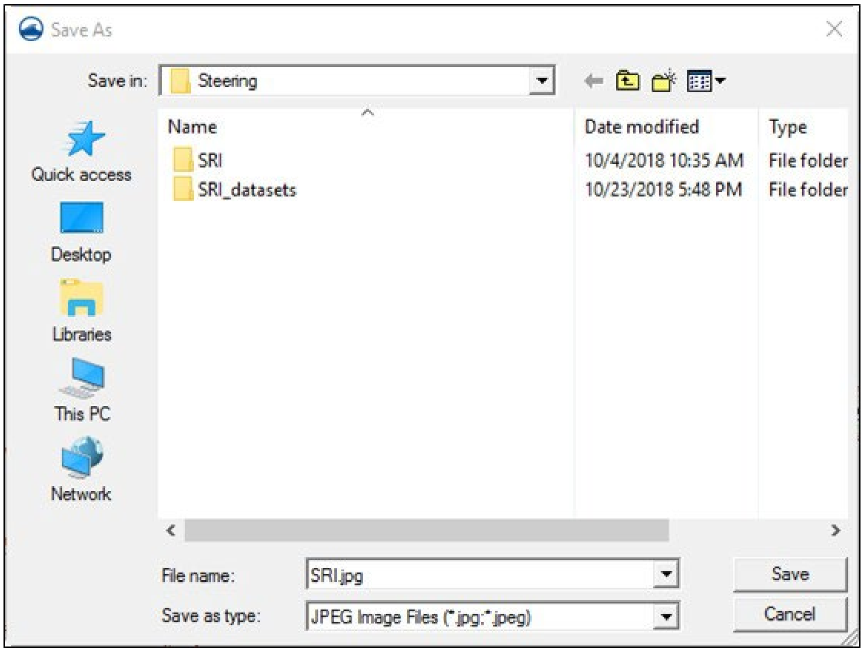


Figure 27. SMS Save As window.

An image file will be saved along with a projection (*.prj) and world (*.wld) file. The projection file contains the horizontal projection information and the world file contains the coordinates of the image origin, the rotation angles (zero for SMS), and pixel size. The image file can easily be loaded and plotted in SMS.

Editing and running the Matlab script

The matlab script below is just an example to show the basic steps in plotting the sediment transport roses in Matlab. For each application, the file names, directories, station coordinates, and settings are needed.

```
%Plots Sediment Transport Roses
clear all
%1. Obtain Observation Point coordinates
tel = Cms_read_tel('..\Hands-on\Flow_Shark.tel');
cosang = cos(tel.angle*pi/180);
sinang = sin(tel.angle*pi/180);
%2. Read transport file
```

```
[qtotal, tel_xx, tel_yy] = read_sed_flux('..\Hands-on\Flow_Shark_trans.sp');
%3. Read and plot background image
A = imread('..\Hands-on\Image_Shark2.jpg');
wld = load('..\Hands-on\Image_Shark2.wld');
xi = wld(5) + (0:size(A,2)-1)*wld(1);
%SMS images are NOT rotated
yi = wld(6) + (0:size(A,1)-1)*wld(4);
imagesc(xi,yi,A) set(gca,'ydir','normal','Nextplot','add')
%4. Plot roses, Rotate transports from local grid angle to world
for k=1:fix(size(qtotal,2)/2)
  qte = qtotal(:,2*k); %East
  qtn = qtotal(:,2*k+1); %North
  [sumx,sumy] = sumdirbin(qte,qtn);
  x = ones(length(sumx),1)*xsta(k);
  y = ones(length(sumx),1)*ysta(k);
  h = vecplot(x,y,sumx,sumy,'marker','^',...'lengthscale',10,'linewidth',2,...
'MarkerFaceColor','k','MarkerSize',3);
end
axis([1.9018    1.9257  1.4979  1.5132]*1e5)
xlabel('X, State Plane, m'), ylabel('Y, State Plane, m')
return
```

The subroutine sumdirbin sums the transport rates over directional bins. Note that to integrate over time the values should also be multiplied by the output interval. However, because the vectors are only for plotting purposes, this step is not necessary here. The Matlab script for summing the transports in directional bins is provided below:

```
function [binx,biny] = sumdirbin(u,v,binsize)
% [binx,biny] = sumdirbin(u,v,varargin)
% Sums the vector u,v in directional bins
% written by Alex Sanchez, USACE
if nargin<3
  binsize = 22.5; %deg, default bin size
end
u = u(:); v = v(:);
mag = sqrt(u.^2+v.^2);
binlim = (-binsize/2:binsize:360-binsize/2)'; %limits
bincen = (0:binsize:360-binsize)';              %centers
ndir = length(binlim);
angle = atan2(v,u)*180/pi;
angle = mod(angle,360)+0.00001;
angle(angle==0) = 0.00001;
binmag = zeros(ndir-1,1);
for k=1:ndir-1
  ind = (angle>binlim(k) & angle<=binlim(k+1));
  binmag(k) = sum(mag(ind));
end
binx = binmag.*cos(bincen.*pi/180); biny = binmag.*sin(bincen.*pi/180);
return
```

The resulting Matlab plot is shown in Figure 28 below.

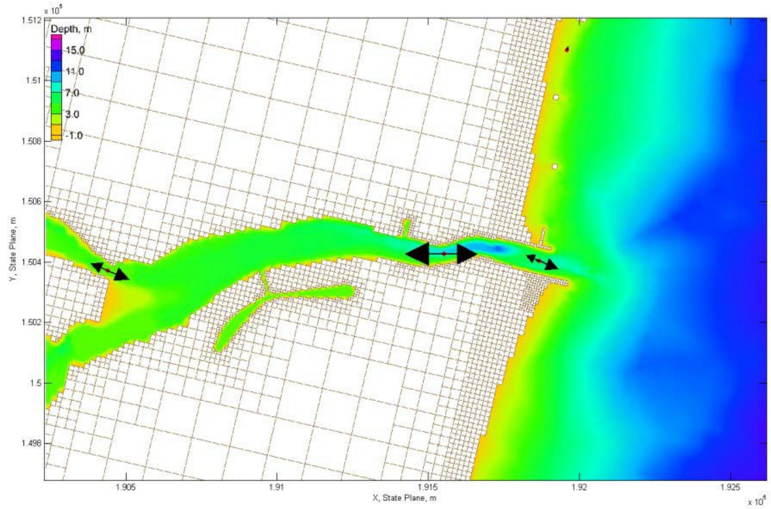


Figure 28. Total-load Sediment transport rates for Shark River Inlet.

Calculating Sediment Fluxes across Observational Arcs

In the CMS, sediment transport can be calculated across lines or polygons defined by feature arcs and polygons in a post-processing procedure. The total-load sediment transport rates calculated from CMS-Flow are integrated across user-defined boundaries. The sediment transport statistics are calculated from the integrated sediment transport rate. The calculated statistics are the net, gross, positive and negative total-load sediment transport rates and are output in units of either cy/ft or m^3/m.

Calculating summed sediment fluxes over cross-sectional arcs can provide channel infilling rates, nearshore sediment transport rates, a sediment budget through a channel, and boundary fluxes.

- Load the solution file from CMS simulation, *_trans.h5.
- Generate a *New Coverage* under *Map Module* and select *Observation* to set up the map module for drawing an observational arc (Figure 29)

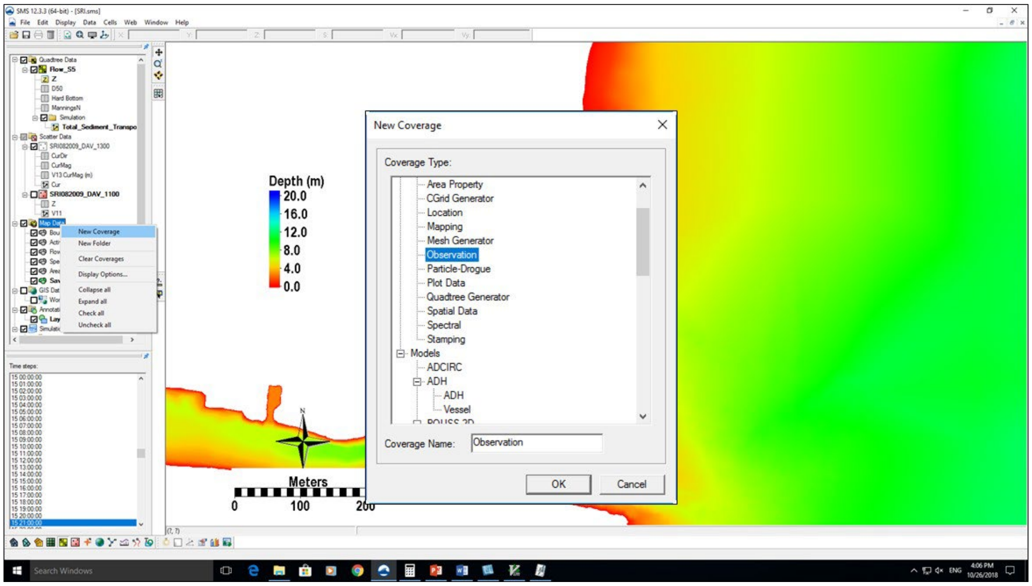


Figure 29. Set up a *Map coverage* type to *Observation.*

- Zoom to the location selected for calculating sediment transport (Example of a cross-shore location shown in Figure 30) and draw the arc using on the *Create Feature Arc* tool  .
- Selecting the *Select Feature Point*  tool will allow modification of the end points of the arc; or the arc can be deleted by selecting with the *Select Feature Arc*  tool.

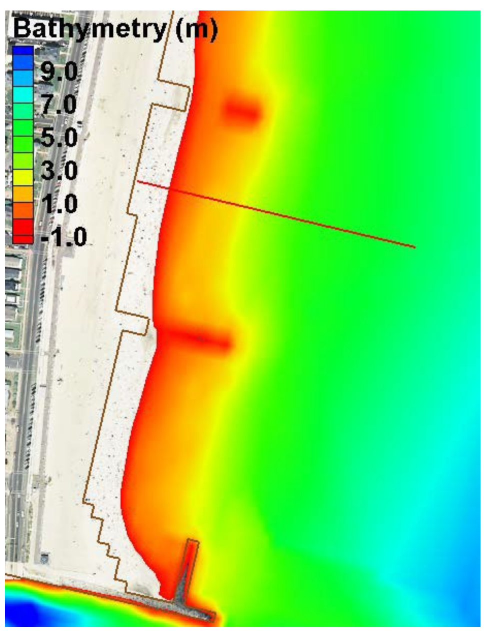


Figure 30. Example of a *Feature Arc* for calculating sediment transport.

Click on the starting and ending points on the grid that define the transect for calculating sediment transport. An example of a *Feature Arc* for calculating longshore sediment transport at Shark River Inlet is shown in Figure 31. In the case of longshore transport, the offshore end should extend beyond the breaker zone and closure depth to capture all, mainly, wave-induced longshore sediment transport. Be sure that the observational arc end points do not touch inactive land cells as this may cause interpolation problems in SMS.

- The positive direction of the sediment transport is defined by the *Feature Arcs* direction. To view the arc direction, click on the *Select Feature Arc* tool  then click on the *Feature Arc* once. The arrows displayed at the beginning and end of the transect indicate the direction of the arc which defines the positive for all fluxes or vectors calculated across it.

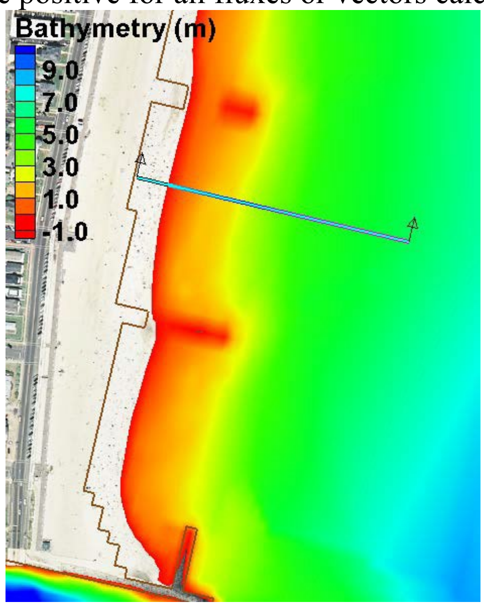


Figure 31. Highlighted Feature Arc with the positive direction shown by the arrows.

- If necessary, the arc direction may be changed by clicking on the Select *Feature Arc* tool and selecting the feature arc by right-clicking it once and selecting the option: *Reverse Arc Direction*.

The sediment transport calculation across a feature arc requires a dataset of the same size and time series length with the constant value of 1. Steps 6-8 describe how to create a new dataset, of any scalar value, within the *Quadtree Data Module*.

- To create this dataset, select the *Quadtree Data Module*, Click on *Data, Data Set Toolbox, Data Calculator.*
- In the *Data Calculator* (Figure 32), select the *water elevation* dataset (1), click on the *Use all time steps* checkbox (2), change the name of the *output dataset* to *ones* (3), double-click on the *water elevation* dataset (4) so that it appears in the *Calculator* window.
- In the *Calculator*, multiply the dataset by zero and then add one (5), select *Compute* (6). The calculation might take a few minutes if the dataset is large. Once the calculation is completed, select *Done* (7), and the output dataset will appear under the *Datasets* window.

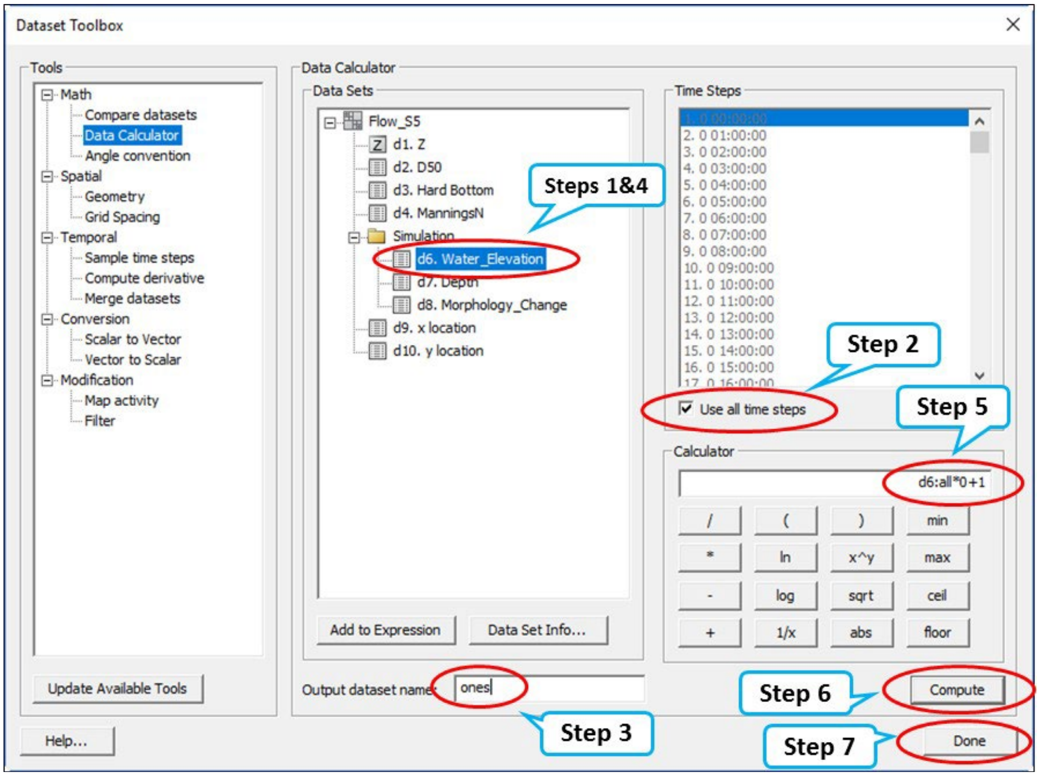


Figure 32. Steps to create a time series dataset of "ones" within the Cartesian grid module.

To obtain the sediment transport statistics, the sediment transport is first integrated spatially to compute a time series of sediment transport across the *Feature* Arc. This procedure is done in the SMS *Plot Wizard.* The sediment transport vector field is interpolated to the points where the arc intercepts cell faces and is then integrated across the *Feature Arc*. Follow each step outlined below:

- Open the *Plot Wizard*, select Time series as the *Plot Type* and then *Next*. In the *Plot Wizard* – Step 2 of 2 window, select *Flux* as the function type.
- Check the feature arc checkbox (Figure 33), select the *ones* dataset for the scalar dataset, and *sediment transport* as the vector dataset, adjust to the correct start and end times, and select *Finish*. (Note that the computation might take a few minutes to tens of minutes, depending on the size of the dataset. This is because SMS must split the vector information, which is computationally slow.)

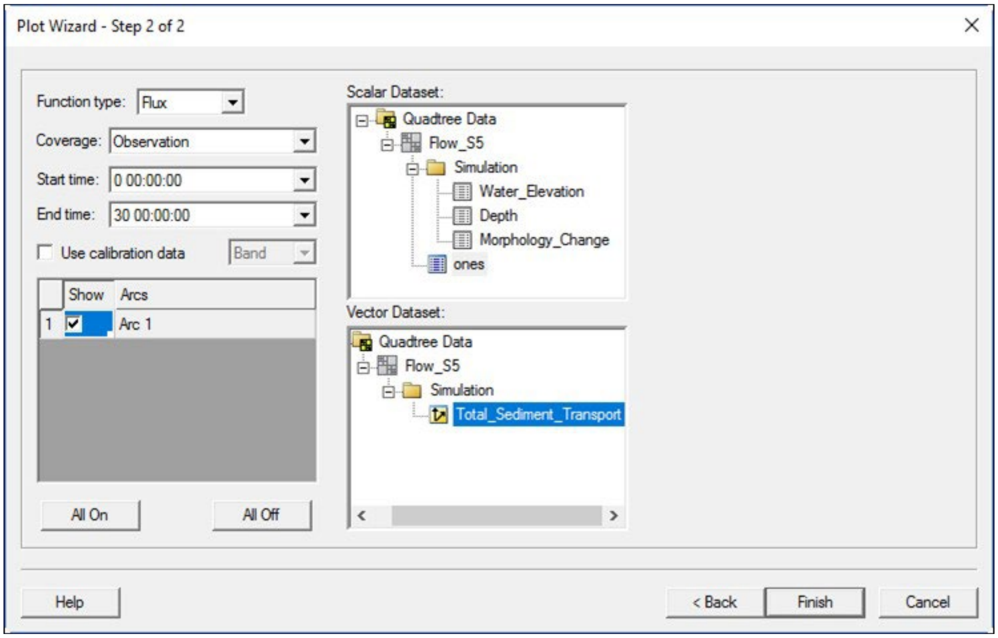
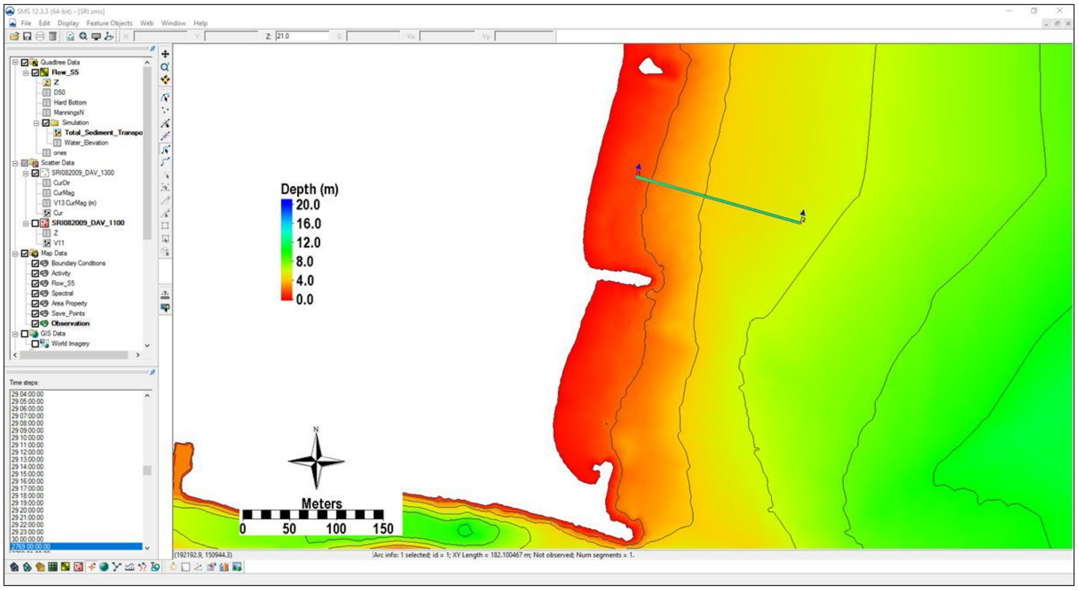


Figure 33. Selecting the correct *Start* and *End Times*, *Feature Arcs*, and *Scalar* and *Vector Datasets* in Step 2 of 2 of the *Plot Wizard.*

Once the time series dataset is created in the form of the graph, the data must be exported for calculations in other software as SMS does not yet have the capability to do so (Figure 34). There are several programs that can process and calculate the sediment transport statistics. One, developed by CIRP, is a simple Fortran executable, though other programs such as Excel can do the same calculations.



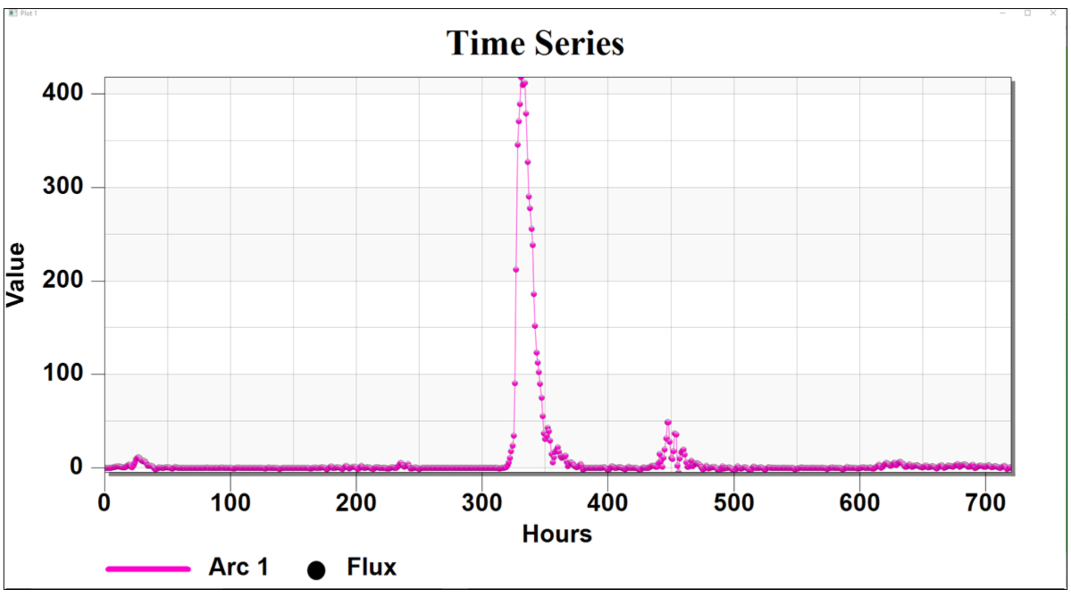


Figure 34. Example time series of integrated sediment transport across a Feature Arc.

The Fortran executable calculates the transport statistics based on the exported time series of integrated (along arc) transport rates obtained in the previous section. The program can be run from a command prompt, a script file or by simply double clicking on the executable and providing the name of the input file. The following explains how to create the input file.

- From the generated graph, export the text data by right-clicking on the time series plot and selecting *View Values.*
- From the *View Values* window, select the data from the table (Figure 35) and copy-paste it to a text editor such as Wordpad or Notepad++. (Note the number of rows in the dataset by looking at the id number of the last row in the *View Values* window.)

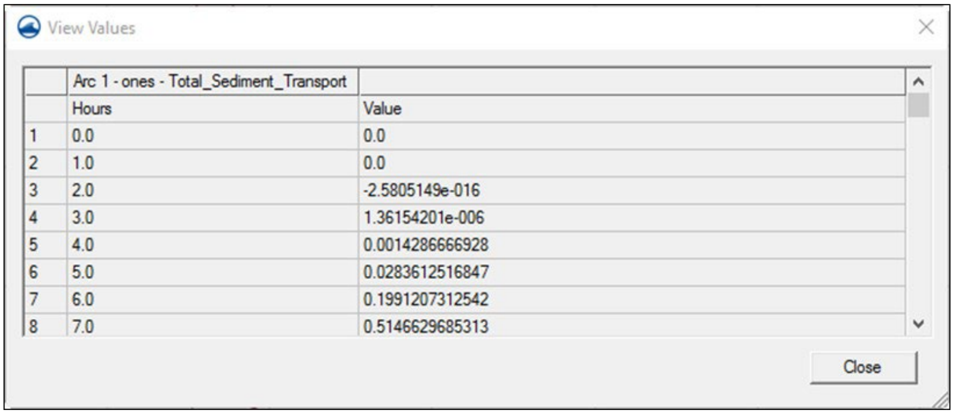


Figure 35. Example of a View Values window in SMS.

- Save the file with the extension **.txt* (Figure 36). The data may also be saved as an *.xys file (SMS xy series file), where the file needs a header (as shown in Figure 36). Make the second entry a file id, the third entry the number of time steps, and the last entry a name for the time series.

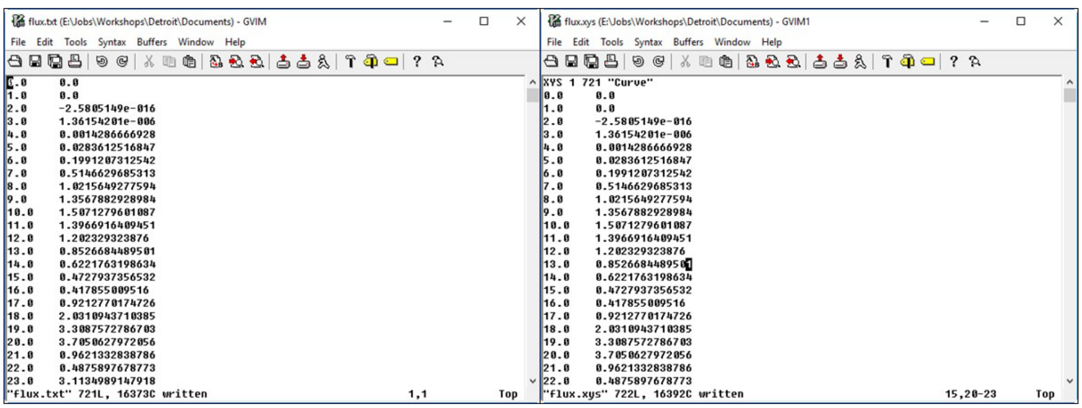


Figure 36. Examples of *.txt (left) or *.xys (right) files with extracted sediment transport time series.

- Double click on the executable named *proc_flux.exe* (Figure 37). The executable does not need to be in the same folder as the input file. If the file is not in the same directory, the path should be specified before the file name. If the path contains spaces quotation marks should be used.

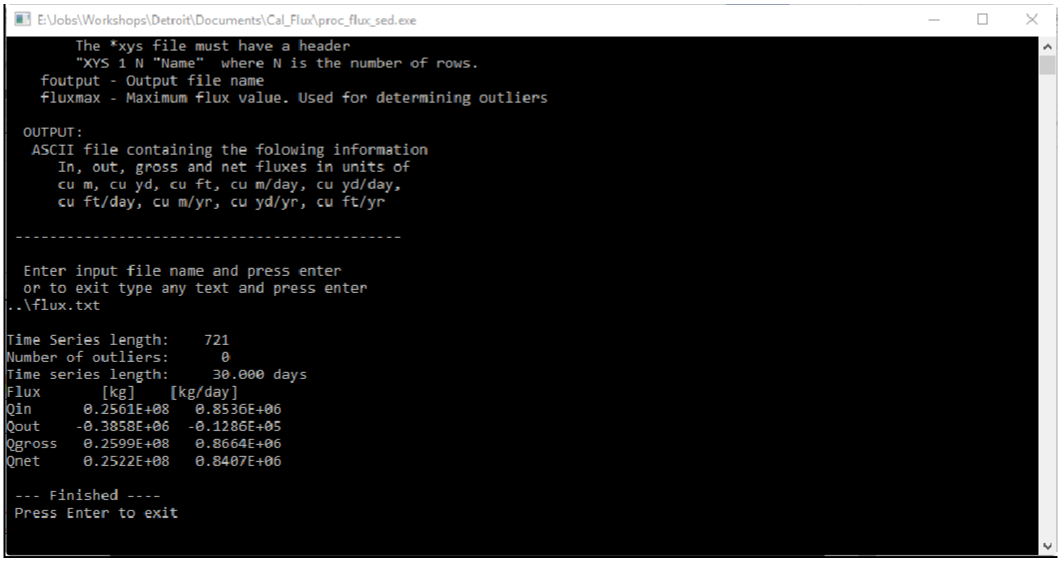


Figure 37. Example calculation of sediment transport statistics using the proc_flux.exe from a command prompt. Note the input file name contains the directory location of the text file.

- Press Enter when the executable is finished. If no output file is specified, the default output file name is Flux_statistics.txt. An example of the output file is shown in Figure 38.

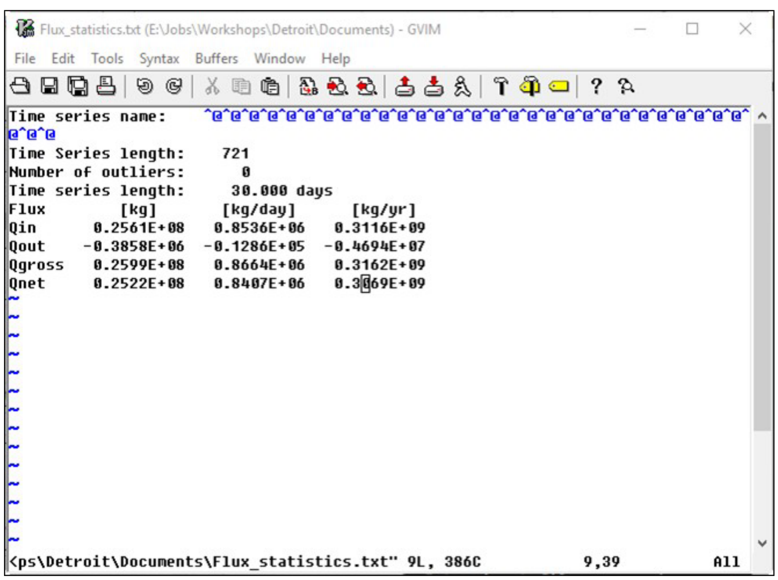


Figure 38. Example output file of proc_flux.exe with sediment transport statistics.

Eliciting sediment transport fluxes from the CMS can contribute toward a check of sediment transport. It is important to note that the sediment transport extracted across an arbitrary arc is not entirely representative of the transport in the greater area. Because the user specifies the arc manually, there is uncertainty involved in capturing the true boundaries of transport processes over the region and should only be used to determine relative fluxes through areas.

Morphology

The results of the CMS calculated morphology change can be visualized several ways, from three dimensional or planar view of the bathymetry, planar view of the volumetric erosion and accretion, and with 1-D cross-sections of the time series. All vector and scalar information can be extracted from points and arcs (and polygons) in SMS. This section will cover the methodology used to post process time series morphology change data.

To plot channel infilling in SMS in a graphical format, the results in the solution file need to be changed from depths (the depth below the datum in which CMS calculates) to elevations. SMS can plot multiple cross-sections, or arcs, against each other, or through time.

Channel Infilling by cross-sections

- Load *the* solution file, if it does not load automatically (*morph.h5).
- In the *Quadtree Module*, click *Data* and *Data Set Toolbox*. The *Data Calculator* should be selected in the *Tools* section as shown in Figure 39.
- Select the *Depth* scalar; under *Time Steps* check the *Use all time steps* on and double-click the *Depth* (highlighted in Figure 39).
- The line under *Calculator* will display "d7:all", add the multiplication symbol (*) and negative 1 (-1) as shown in Figure 39.
- Change the *Output data set name* as shown in Figure 39 so as to distinguish this dataset from the original *Bathymetry* output, click *Compute*, and after the scalar set appears in the SMS window (Figure 40), click *Done*. (Note: values for the contours may need to be adjusted to view elevation range.)

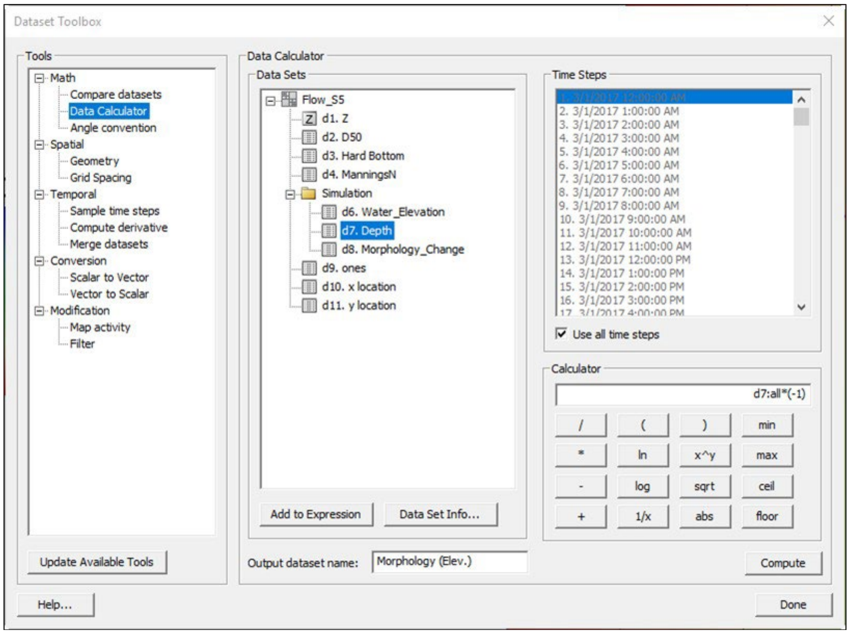


Figure 39. Data Calculator Options in the Dataset Toolbox.

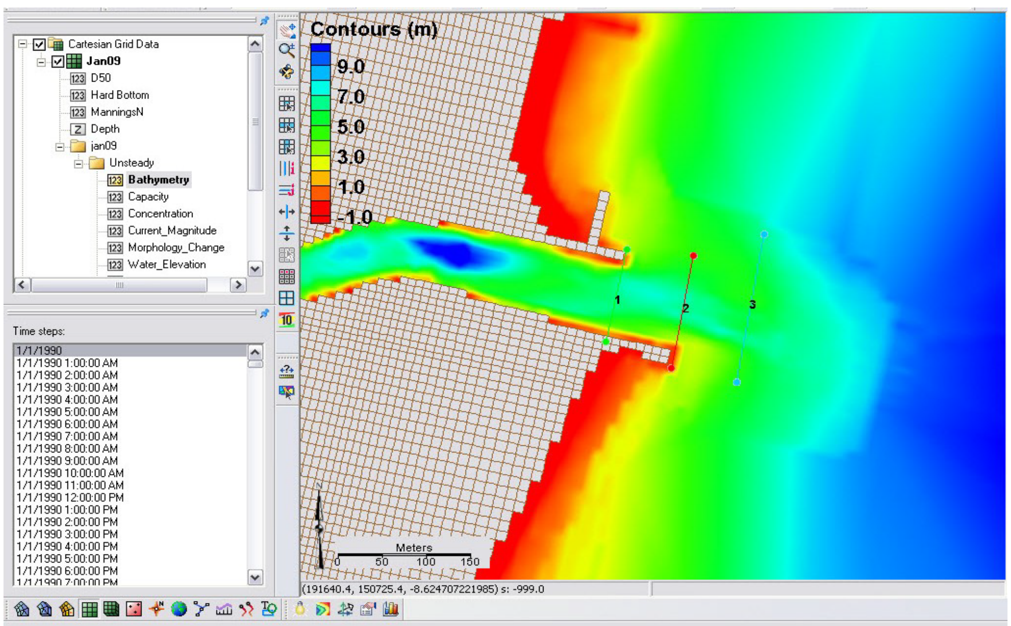
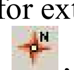
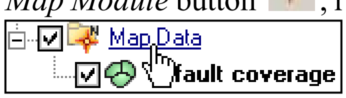


Figure 40. Display of calculated bathymetry now converted to elevations. Note the legend has been modified to display the new contours.

- Create feature arcs for extracting the time series data by switching to the *Map Module* button , right-clicking *Map Data* 
, selecting *New Coverage, Observation,* and naming the coverage (Observation1 here).

- Click the *Create a Feature Arc* button  to generate arcs from which cross sections will be extracted and click on one side of the channel and double-click on the other side to close an arc.
- Click the *Select a Feature Arc*  button and select the generated arc (Note that there is a direction associated with the arc, which determines the sideview of the cross section)
- Select an arc, right-click it, and select *Attributes*, and set the color and name(s) of each arc which are listed together in the *Observation Coverage* options (Figure 41), click *OK*

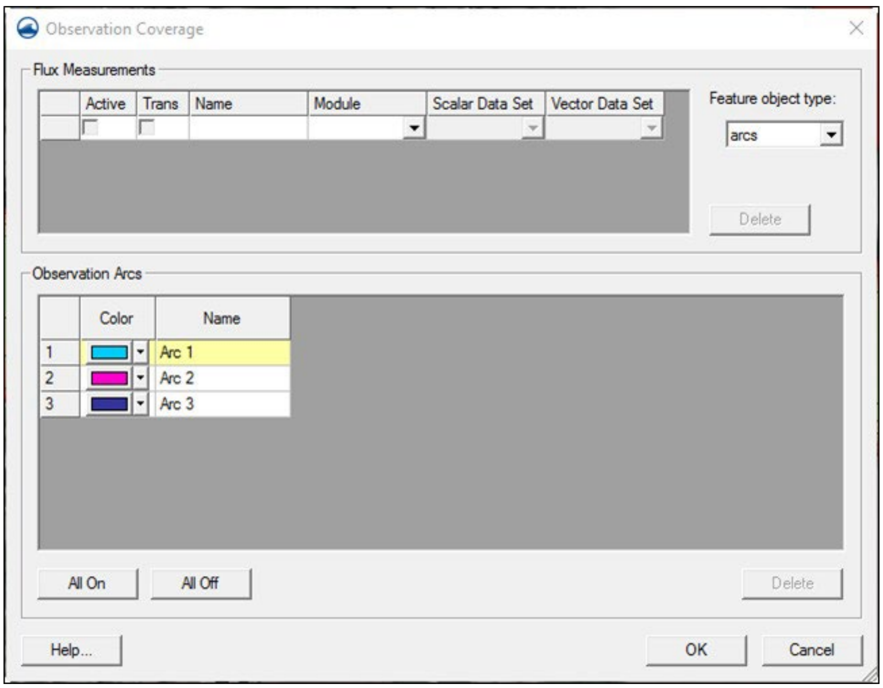


Figure 41. Observation Coverage attributes

- To plot the profiles in SMS, Select *Display*, *Plot Wizard*, and *Observation Profile*, and select *Next.*
- Following Figure 42, under *Coverage* check one arc (though multiple can be plotted), under *Dataset(s)* select *Specified dataset(s)* and select the generated *Morphology* (Elev.) set only, and under *Time step* select *Specified time step(s)* and check on several times spaced apart by at least several days to months, click *Finish.*
- Right-click on the plot (Figure 43) and select *view values*.., and a table will appear with all the plotted distance and elevations given in the graph, (Note this ASCII table can be copied and pasted into *excel*.)

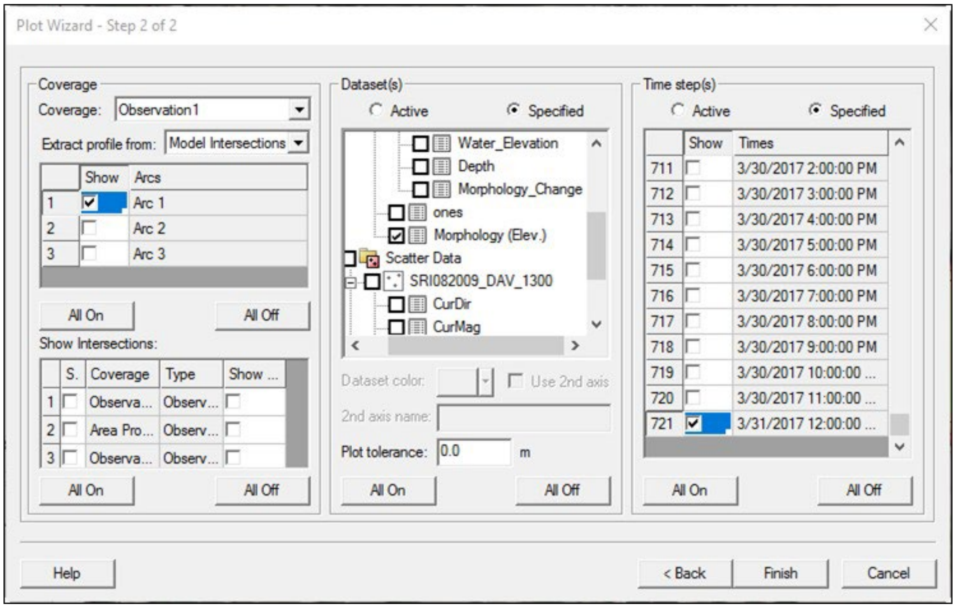


Figure 42. ***Plot Wizard*** **Step 2.**

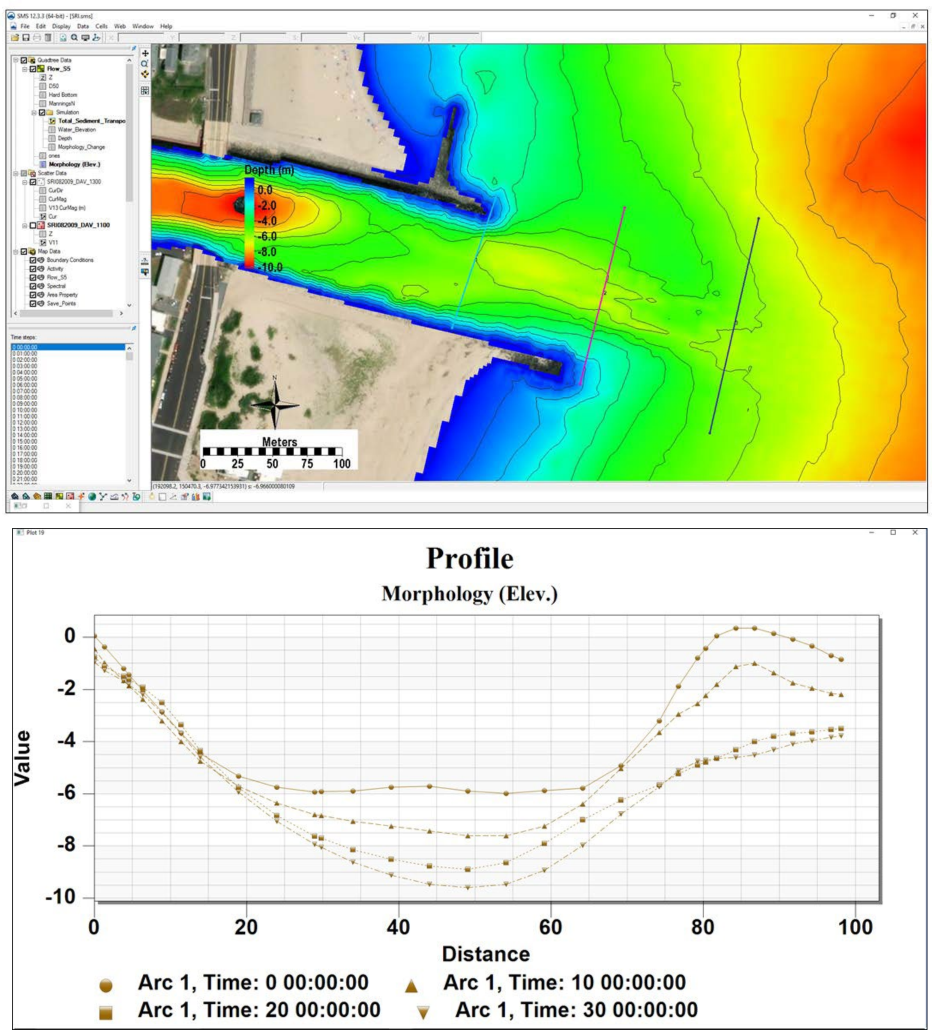


Figure 43. Plotted cross-section.

- To export the data in different formats, right-click on the plot, and select *Export/Print* for several options to export the graph or data from the graph (Figure 44)
- To add more profiles or time-series, right-click on the plot, and select *Plot Data* to bring up the original options from Figure 42.

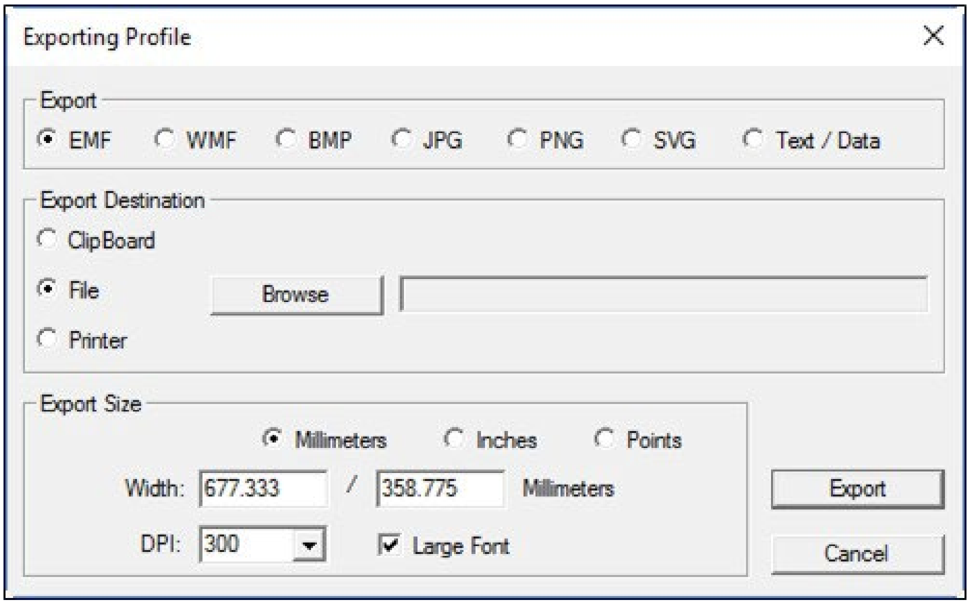


Figure 44. Data Export Options.

Analysis of cross-sectional area and sediment volume changes are the most common approaches to quantifying morphologic change. Another method of quantifying change in a spatial framework is through comparison of the planform aerial change. This method is useful in delineating active zones of transport, typically based on the depth of closure, and can define the upper and lower limits of transport. These results can be used to interpret bypassing areas or zones, however, determining the bypassing pathways should not be based on this analysis alone.

Calibration and Validation

Often, the default user-defined parameters assigned to each cell are not the best representation of the environment. This holds particularly true for areas with substrate bottom different than typical estuarine and littoral sands whether they are marsh, hardbottom, or a rubble pile. The default frictional value (Mannings n, 0.025) must be adjusted to better represent the hydrodynamics in these areas. An example in Shark River Estuary might be the very shallow, north and south up-river reaches that are dominantly marsh platforms. These areas flood and dry within the tidal range, but with a

> "…that a much too great generality often obscures more than it enlightens and leads to calculations so intricate that it becomes extremely difficult to deduce from them consequences even for the simplest of cases…."
>
> - Leonhard Euler

higher friction than an open sand substrate, and therefore the frictional factor must be adjusted appropriately.

Additionally, flow in channels that are influenced by structures, such as bridge pilings, piers, or walls, must also include some modification to the frictional factor. For a seawall or shear wall, this is represented when the cell is designated a "land cell"; however, if your structure is much smaller than cell size an alternative option would be to increase the friction factor to imitate the frictional forces of that structure. This was done in an earlier section describing the bridge pilings located at three bridges crossing the main channels are Shark River Inlet.

Checklist

Below is a checklist for common issues that should be addressed during the initial calibration process.

1. Check the Bathymetry. The most common mistake in bathymetry is improper conversion to the common vertical datum. This will result in jagged triangulation within the grid, or a large increase or decrease that resembles a morphologic feature such as a shoal but is in fact just poor datum control. Sometimes the horizontal datum may be off by a small margin. For example, many Federal Agencies still collect spatial data in NAD27 instead of the NAD83, and from a large scale these can easily be confused.
2. Grid dimensions. Does the grid cover the reaches it needs to support proper tidal prism, littoral range, and cover enough area to provide room to calculate large wave and hydrodynamic gradients and to avoid boundary interference for interior calculations?
3. Timing. Setting up the proper dates and times in the model control is crucial in calibration. Common mistakes include missing the measured time period in the forcing or the model period, not coordinating all forcings, and improperly interpreting the SMS output date and time intervals. Also, beware the differences between local time and Global Time (GMT/UTC).

Statistics

Statistics can be calculated on the equidistant measured and calculated profiles. There are many statistical approaches to compare the measured and calculated data for hydrodynamics. Of those approaches Correlation Coefficient, Root Mean Square Error (RMSE), Normalized or Relative Root Mean Square Error (NRMSE) are commonly used for interpreting goodness of fit for hydrodynamic measurements. The Brier Skill Score and Correlation Coefficient methods are often used in evaluating bathymetry change over transects or morphologic differences such as volumetric change for morphologic features.