8. Post-processing Results

8.1 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 good 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. Both fixed and roaming data can be assessed over a time series, where boat-mounted data are collected at specified locations over time intervals. However, roaming data that were collected continuously must be assessed as a function of location in three-dimensional space.

With good forcing data and accurate corresponding datums, the final check before hydrodynamic calibration can be achieved is the accuracy and detail of the geomorphic parameters such as bathymetry, sediment characterization, and structure definition. The most important of these is bathymetry, which can have several issues which are typically associated with quality control, density, interpolation, and final 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 8.1.1). 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 prism to a shallow bay platform.

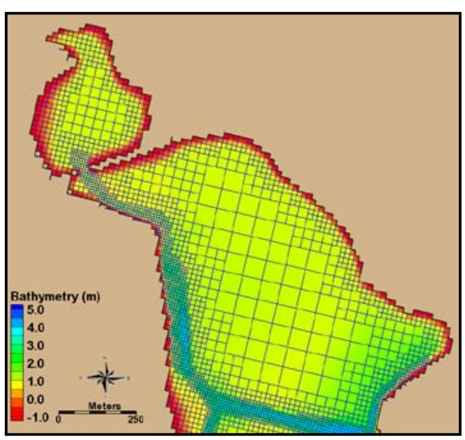


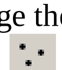
Figure 8.1.1. Northern portion of the Shark River estuary.

Figure 8.1.2 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 8.1.2. Location of NOAA (red) and USGS (blue) water level gauges.

- Importing water level measurements in to SMS for comparison:

  1. Solution files from both an explicit and implicit circulation model are provided in this section's folder. Open the cmcards file in the *Implicit_Calibration Run* folder, which should bring the solution file in automatically. (This can serve as a backdrop for locating the gauge location in SMS.)

  2. Change the map *Default Coverage* type to *Observation*, and add a *Feature Point*  to the approximate location of the tide gauge shown in Figure 8.1.2.

  3. Select and right click on the *Feature Point|Node Attributes*, and the *Observation Coverage* window will pop up (Figure 8.1.3). Under *Measurements*, select the drop down menu under *Module*, and change it to *2D Cartesian*. Change the *Dataset* to *Water Elevation*. Select *Trans* (transient data), and the *Time Series* menu for the observation point and select *Options…*

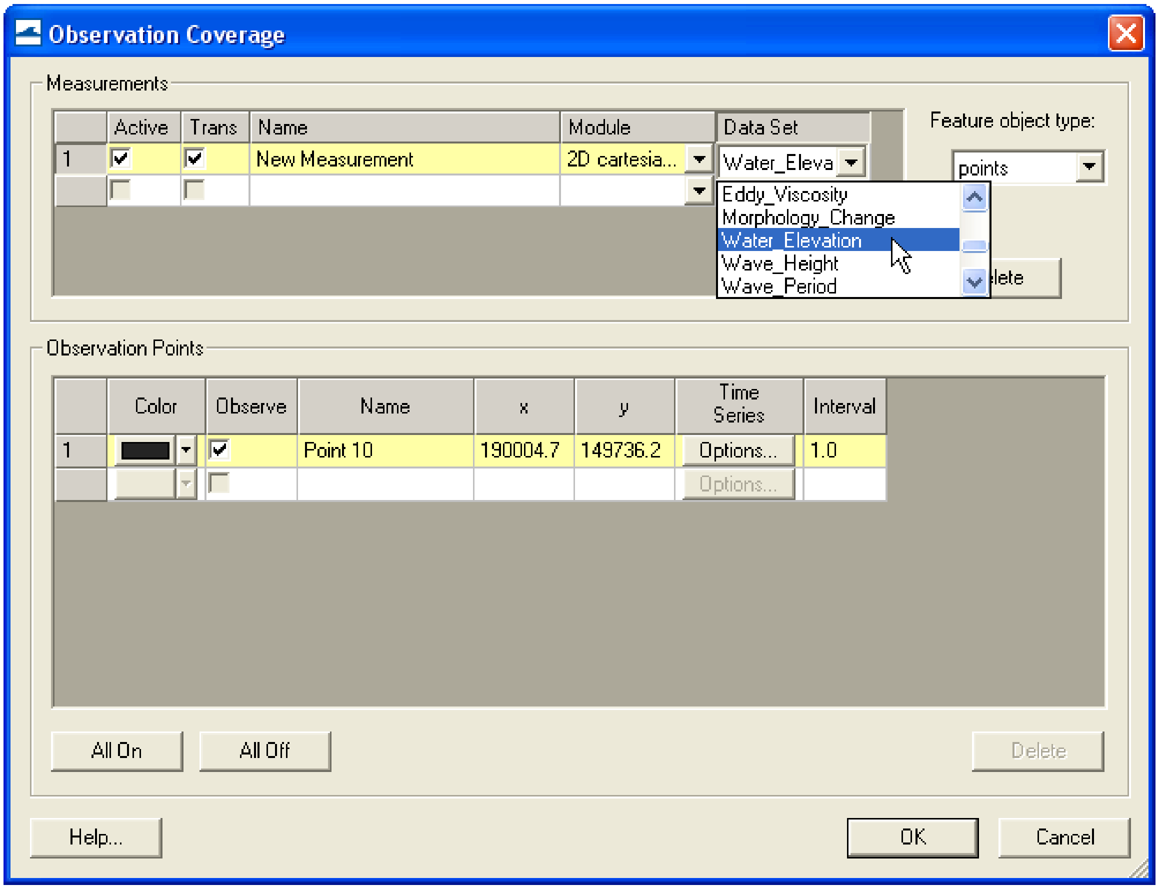


Figure 8.1.3. Setting up the measured time series data within the Observation Coverage window.

4. 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.

- Water level data plotting from Feature Points:

1. Time plots can be compared within SMS through their software plotting package. To create a plot 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 8.1.4).

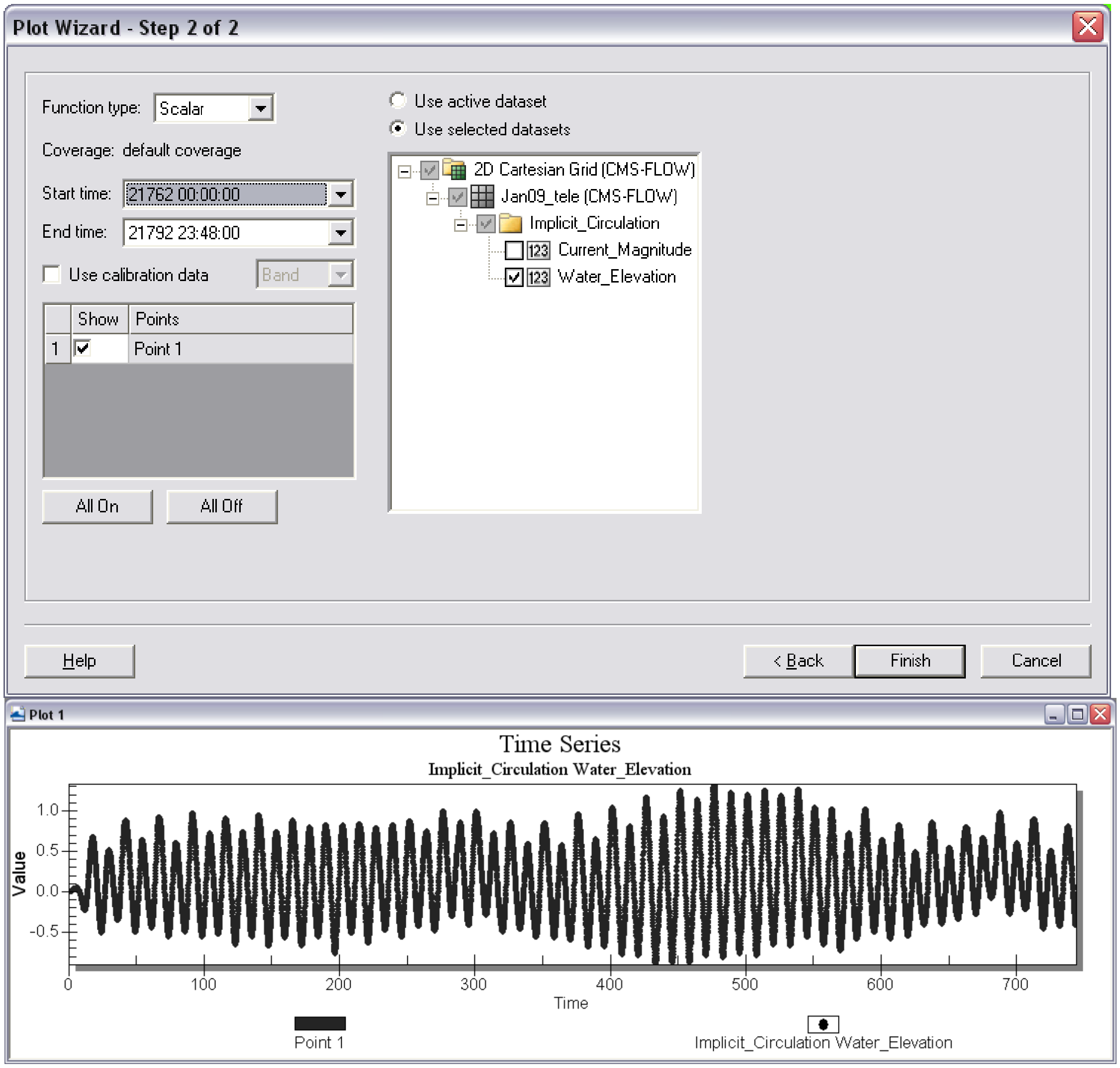


Figure 8.1.4. Plot Wizard for time series data (top); Time Series plot of Water Elevation (bottom).

2. The plots can be updated by right-clicking the graph window and selecting *Plot Data*. Also, the time frame can be modified.

3. 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.

4. 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 8.1.5). Export.

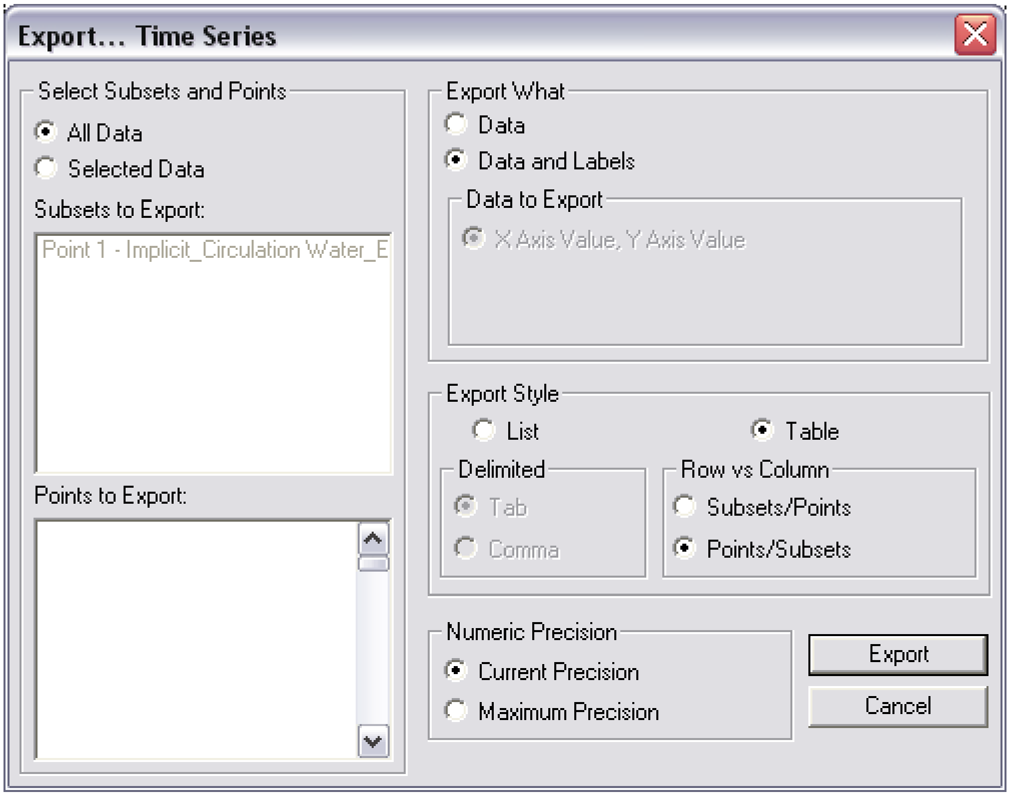


Figure 8.1.5. Export Wizard for plotted data.

5. 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 8.1.6). Because the Sandy Hook gauge is located 30 km north of Shark River, the calculations have a slight phase advance in comparison to the measurements because the tidal wave propagates from north to south on this coast. The ocean gauge typically leads the bay gauge by 20-30 min.

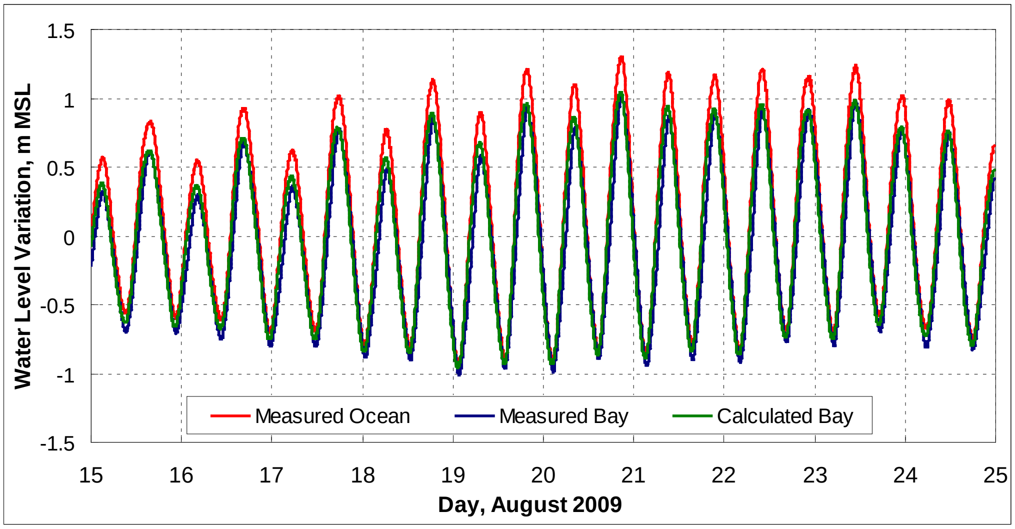


Figure 8.1.6. Observed time series of water level at Sandy Hook and Belmar ("Bay") and calculated water level at Belmar.

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:

    1. To create a 30 minute dataset of water levels from the original 6 minute output, go to *Data*, *Data Calculator*, and under the *Tools* section, select *Temporal*, *Sample Time Steps* (Figure 8.1.7).

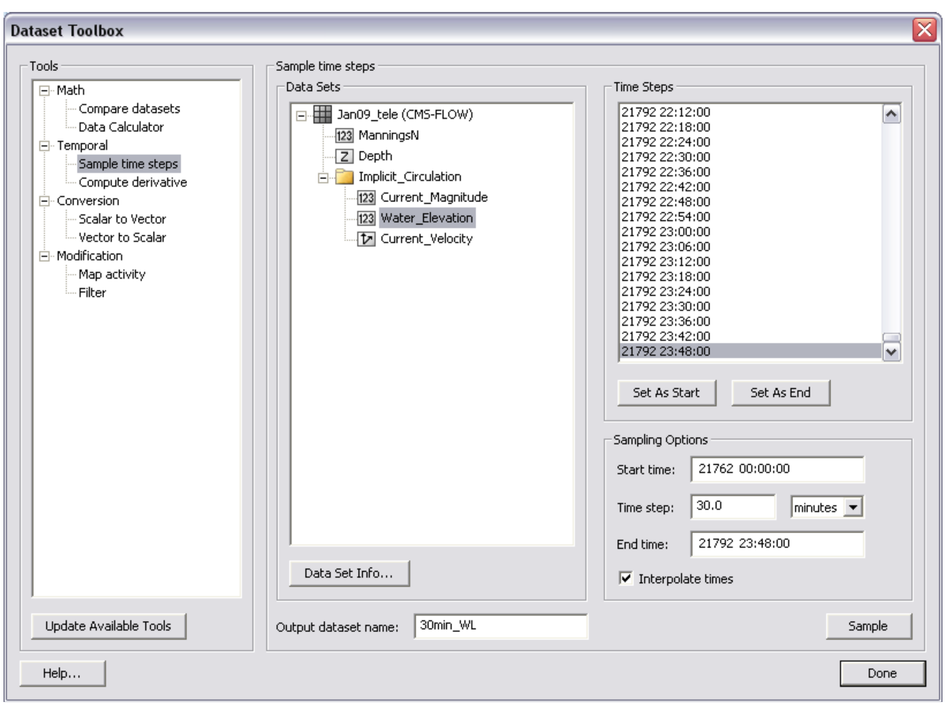


Figure 8.1.7. *Sampling time steps* of the *Water_Elevation* scalar dataset in the *Dataset Toolbox*.

2. Select the *water level* dataset under the solution datatree, and create an *Output dataset name* at the bottom.

3. 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), select *Sample*.

Of the many types of velocity measurements, those taken for Shark River Inlet in August 2009 were D-ADCP transects, or 2-D profiles along transects. Velocity was binned into depth measurements throughout the water column, and these were preprocessed and depth-averaged to produce comparable velocities to those from the CMS run. To display preprocessed measured velocities, they must be converted into an XYZ format with additional data (velocity and direction, or Vx and Vy) 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:

1. Open the provided file in the section folder SRI082009_DAV_1300.GIS, and select *Use Import Wizard* (An excel version of the text file is also provided in the folder). Select *space/tab* deliminated if not already checked and click *Next*, and select the X, Y, Z, Vector X and Vector Y columns (Figure 8.1.8). Be sure that the Header of the Vector X & Y columns is the same. Change the headers so they are the same (Vector), otherwise this incurs an error when importing in SMS.

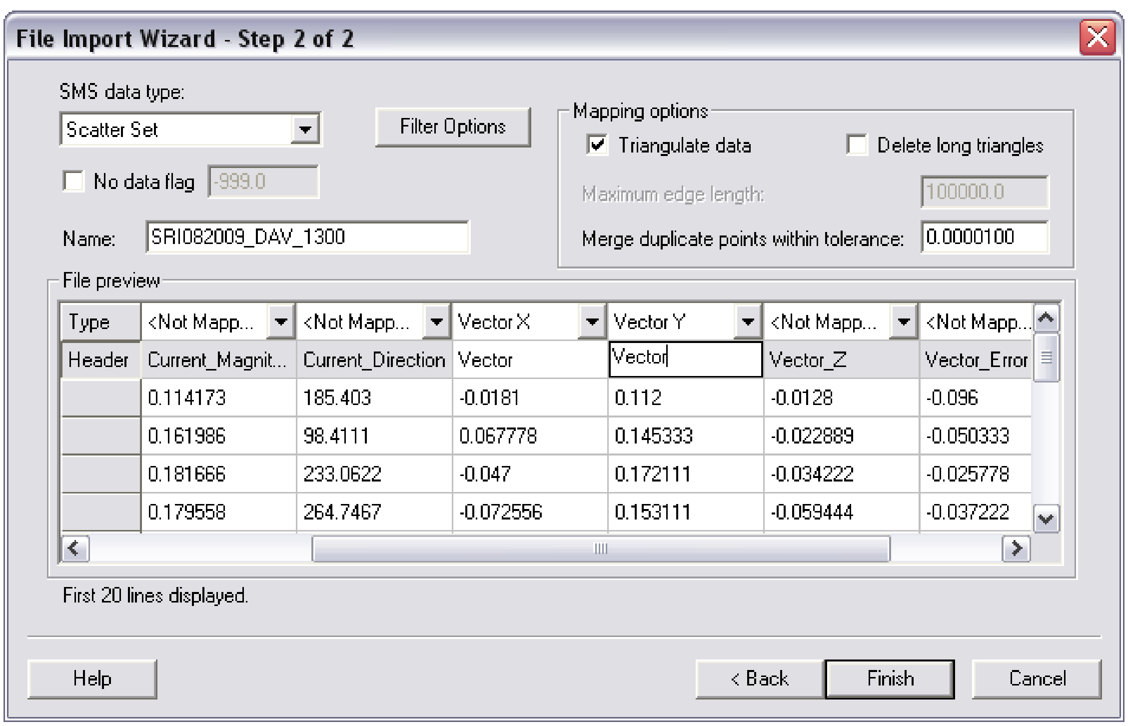


Figure 8.1.8. *Sampling time steps* of the *Water_Elevation* scalar dataset in the *Dataset Toolbox*.

2. Change the vectors back to scalar and direction by right clicking on the *Current_Magnitude* scalar dataset and selecting *Vectors to Scalars*, *Magnitude and Direction*, and *OK*.

   The scatter set should have a vector dataset for current listed below depth, and the scalar magnitude and direction datasets below these (Figure 8.1.9). Selecting these scalar datasets will display the quantities of each point, such as velocity magnitude or water depth, and will illustrate the contouring between the points of the transects.

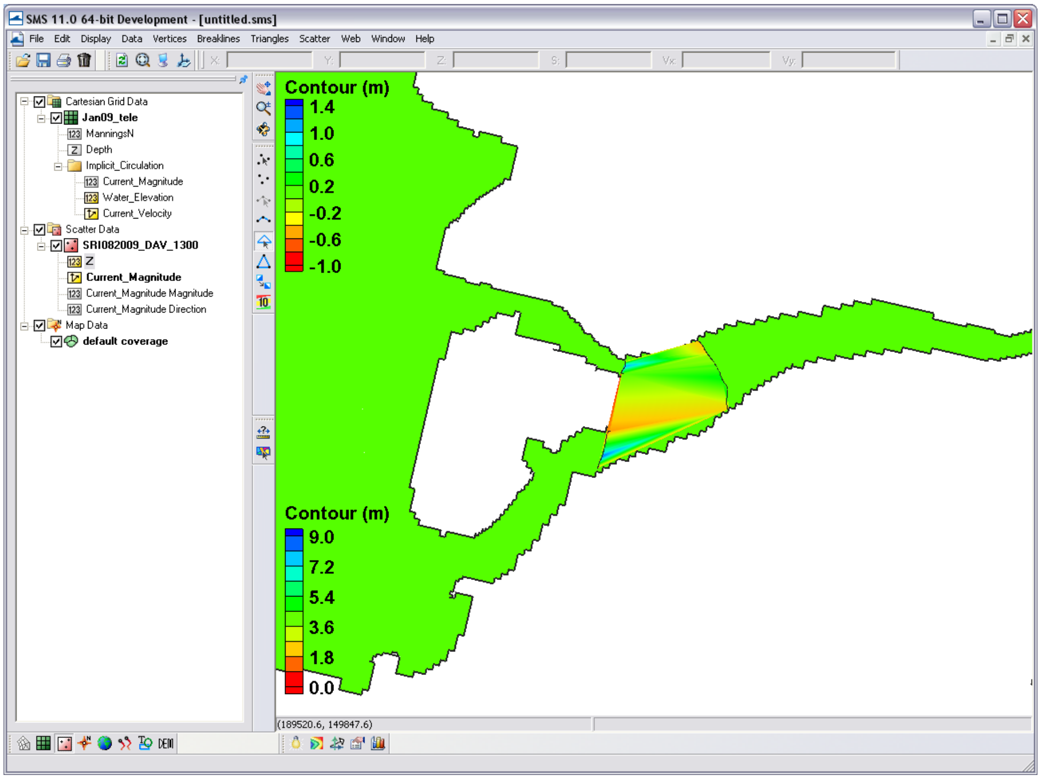


Figure 8.1.9. Imported velocity transect file SRI082009_DAV_1300.GIS.

3. To display the velocity as arrows with magnitude and direction, go to *Display Options*, the *Scatter* tab, and check *Velocity Vectors* on. Modify the vector display under the *Vectors* tab by setting the Vector Display to *At each node*, setting the *Z-offset* to 10.0, and the *Arrow Options Shaft Length* to *Scale Length to Magnitude* (Figure 8.1.10).

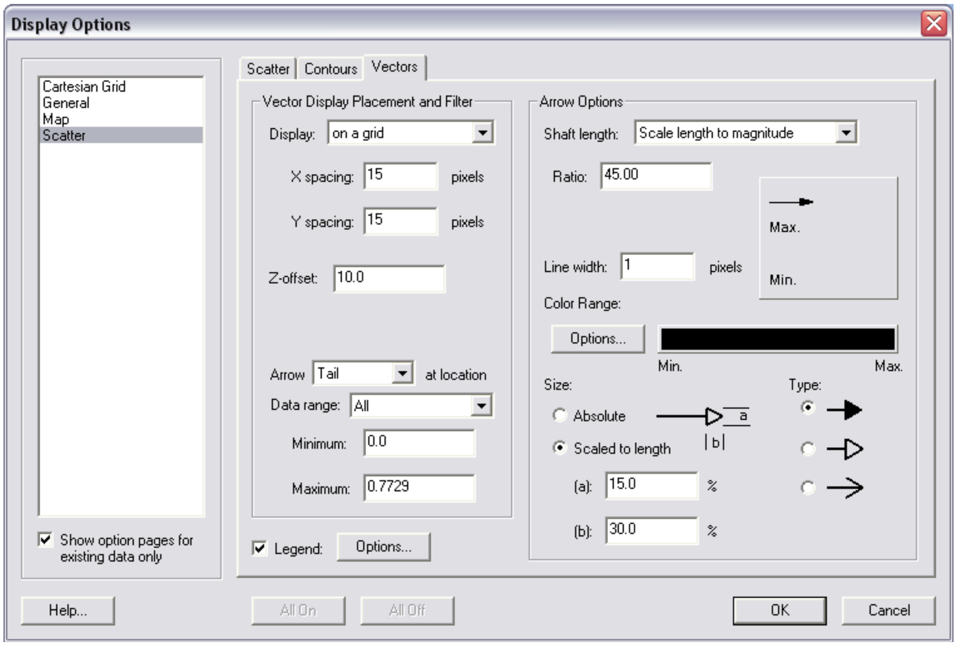


Figure 8.1.10. Vector Display options for scatter points.

4. 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 8.1.11.

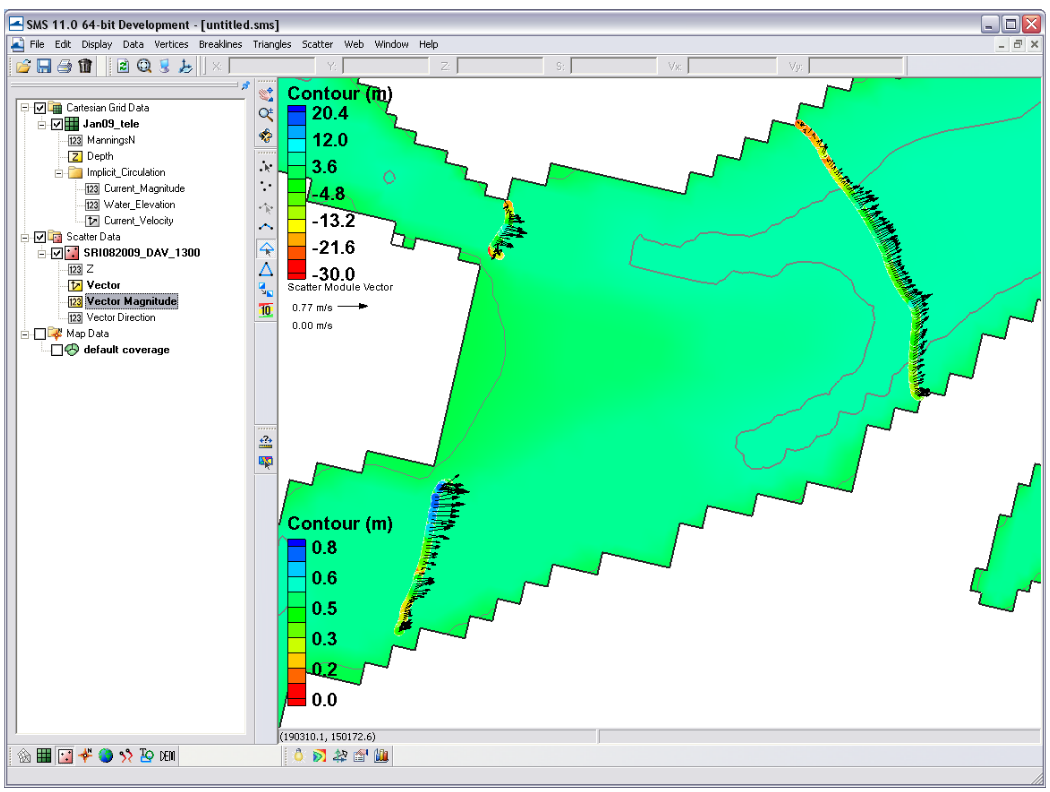


Figure 8.1.11. Color options turned on for scatter points.

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. 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:

    1. Following the above section, the current velocity file is displayed similar to Figure 8.1.12. Draw three observation arcs over each of the three transects.

    2. 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 8.1.12 illustrates everything selected for plotting the measured and calculated data for the specific time the transect was actually measured.

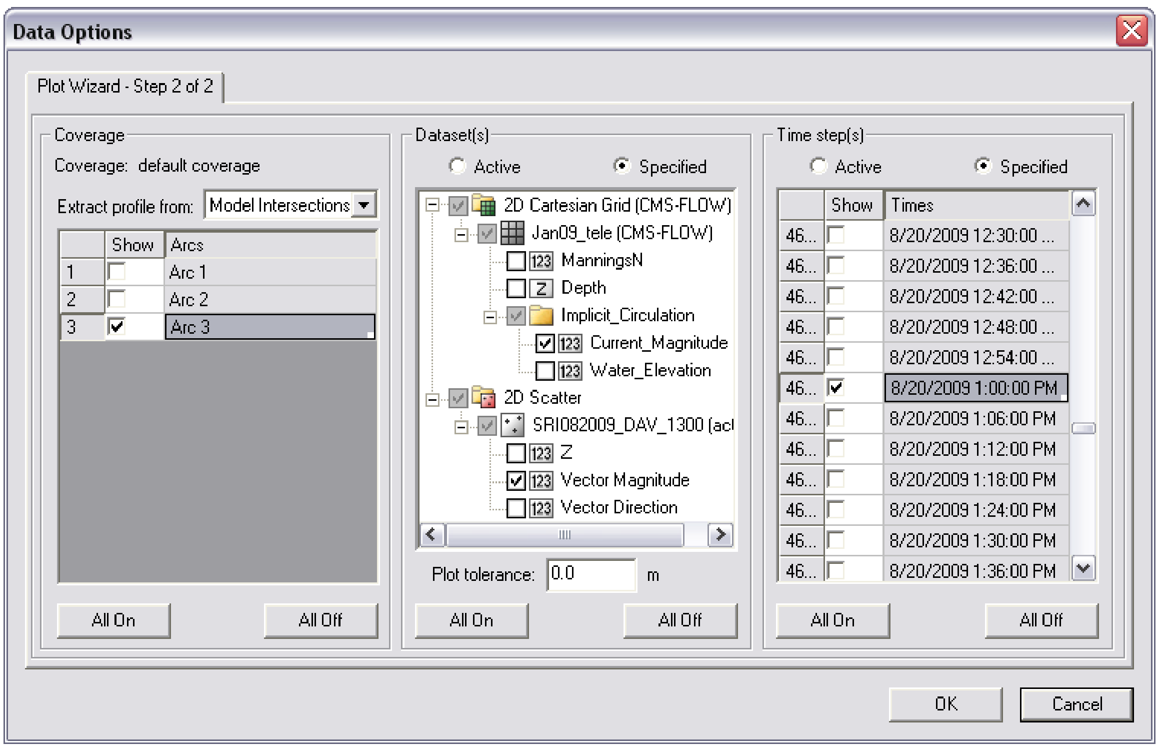


Figure 8.1.12. Plot Wizard data options for displaying measured and calculated data from an observation arc.

3. 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.

4. The resultant plot is shown in Figure 8.1.13, 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.

5. To adjust the points distance output from both the calculated and measured, select and right click on an arc, select *Redistribute Vertices*, and set the vertices to a *Specified Spacing* such as 10 meters. Right click on the plot, and select *Plot Data*, to modify the selection/extraction of point data from *Model Intersections* to *Points and Vertices*.

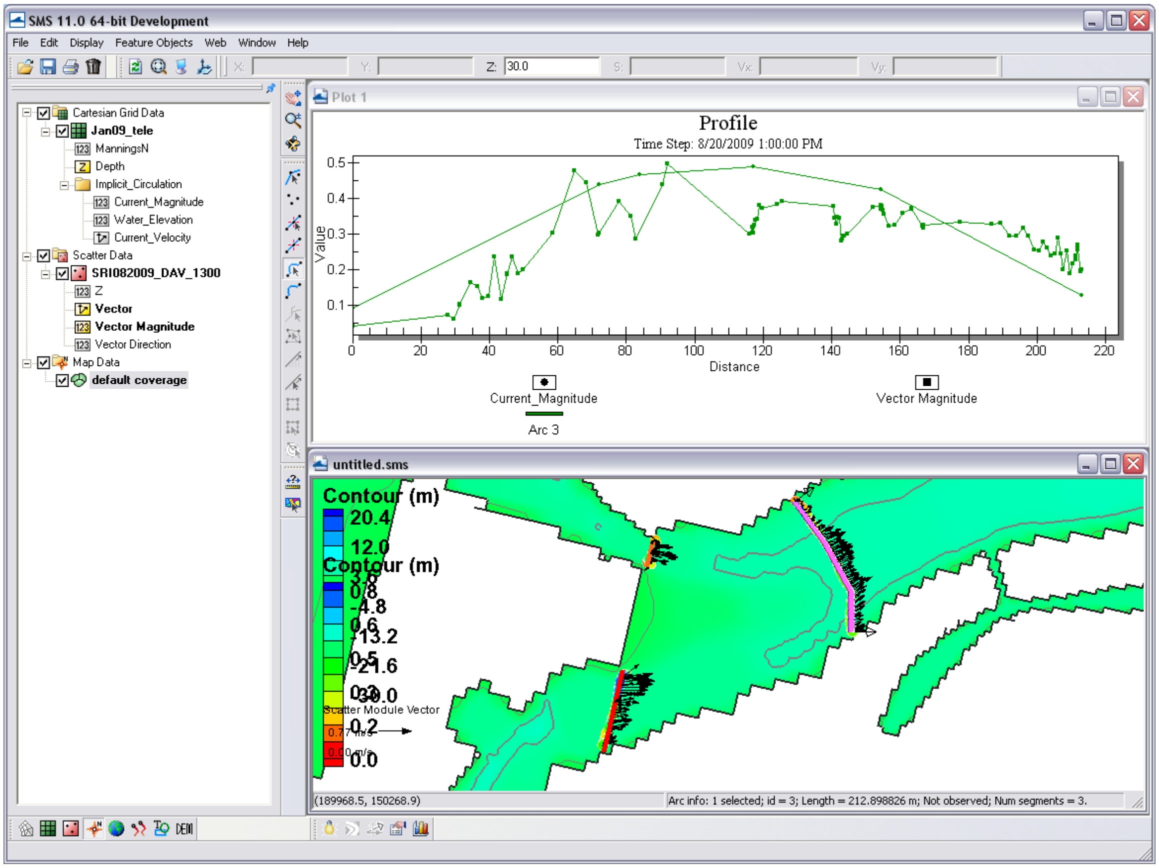


Figure 8.1.13. Plotted measured and calculated data from the larger observation arc to the East. Note that the positive direction reflects the observation arcs direction.

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 listed below in Table 8.1.1, RMSE, Correlation Coefficient, and RMAE 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.

Table 8.1.1. Common equations for calculating statistics.

Root-Mean-Squared Error	$RMSE(x,y) = \sqrt{\left\langle (x-y)^2 \right\rangle}$
Mean-Absolute Error	$MAE(x,y) = \left\langle \lvert x-y \rvert \right\rangle$
Relative-Mean-Absolute Error	$RMAE(x,y) = \frac{MAE(x,y)}{\lvert x \rvert}$
Correlation Coefficient	$R = \frac{\langle xy \rangle - \langle x \rangle \langle y \rangle}{\sqrt{\langle x^2 \rangle - \langle x \rangle^2}\sqrt{\langle y^2 \rangle - \langle y \rangle^2}}$
Brier Skill Score	$BSS(x,y) = 1 - \frac{\left\langle (x-y)^2 \right\rangle}{\left\langle (x-x_0)^2 \right\rangle}$

8.2 Post-processing CMS-Wave Results

This section describes three FORTRAN codes, **concatwav.exe**, **pkcmswave.exe**, and **pkcmswave1.exe** to assist in handling the CMS-Wave output information. Three MATLAB codes, fig1sets.m, fig2sets.m, and fig4sets.m, are provided to plot and compare calculated wave parameters and/or measured data (NDBC or CDIP buoys).

An example is provided as a simple CMS-Flow explicit steering run of the Shark River application (run 2009.sim and 2009.cmcards for a total of 12-hr simulation with 3-hr steering interval – a10-min run time with 3-processors assigned to CMS-Flow).

- **concatwav.exe** - this FORTRAN code merges all individual cycle *.wav files from a CMS-Flow explicit and CMS-Wave steering (coupling) run into one single *.wav. Running **concatwav.exe** requires enter a text file that lists all or more individual *.wav files in sequential order (one-column format) – a sample text file concat.dat is provided that has included the first 200 cycle *.wav filenames. Users can avoid this code if a std.dat file (a modified copy of standard *.std to overwrite all individual *.std files generated in the steering process) is prepared with the 7th control parameter IWET set to -2 before running the steering. IWET = -2 will output a total.wav file include all individual cycle *.wav. In the present time, users should manually prepare the std.dat and set IWET = -2 for this option. In the SMS11 Beta and future SMS release, users can simply set IWET = -2 in the standard *.std (the std.dat is no longer required).

    1. Under the DOS window, run **concatwav.exe** in the folder with all steering cycle *.wav files

    2. Responding to the on-screen input, type the name of a text file that lists all or more individual *.wav files in sequential order, or simply type the sample file ‘concat.dat’ that includes the first 200 cycle *.wav filenames (by using ‘concat.dat’, the output total.wav will include up to a maximum of 200 cycle *.wav) – see Fig 8.2.1 to merge all 5 cycles *.wav into one total.wav file.

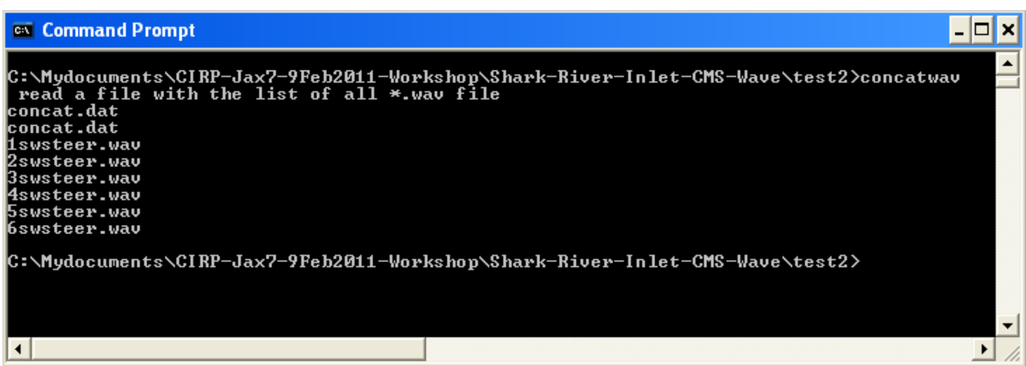


Figure 8.2.1. Run concatwav.exe.

- **pkcmswave.exe** – this code will extract the calculated wave height, period and direction information from *.wav for a single cell in the model grid. The output file is 'oneline.dat' that lists the extracted wave information (in sequential order) using the Wave Information Studies (WIS) short format (11-column format: id, year, month, date, hour, sig wave height, peak period, mean period, mean wave direction, wind speed, wind direction) – show all direction in the meteorological convention.

  1. In the DOS window, run **pkcmswave.exe**

  2. Responding to the on-screen input, type the CMS-Wave wave output *.wav filename

  3. Type the year (last two digits of a calendar year), e.g. type '11' for 2011

  4. Type the local shoreline orientation (the CMS-Wave grid y axis) in clockwise polar coordinates (deg, positive from North covering the sea, e.g., 180 deg for St Mary's Entrance, FL/GA, or 360 deg - the wave grid orientation angle in *.sim)

  5. Type a single cell i and j indices (two integer numbers) saving wave parameter information

  6. Type the starting timestamp (default value is 0) and ending timestamp (default is 99999999) for saving output files – see Fig 8.2.2 to run **pkcmswave.exe** to extract wave information at Cell (69, 92) from total.wav.

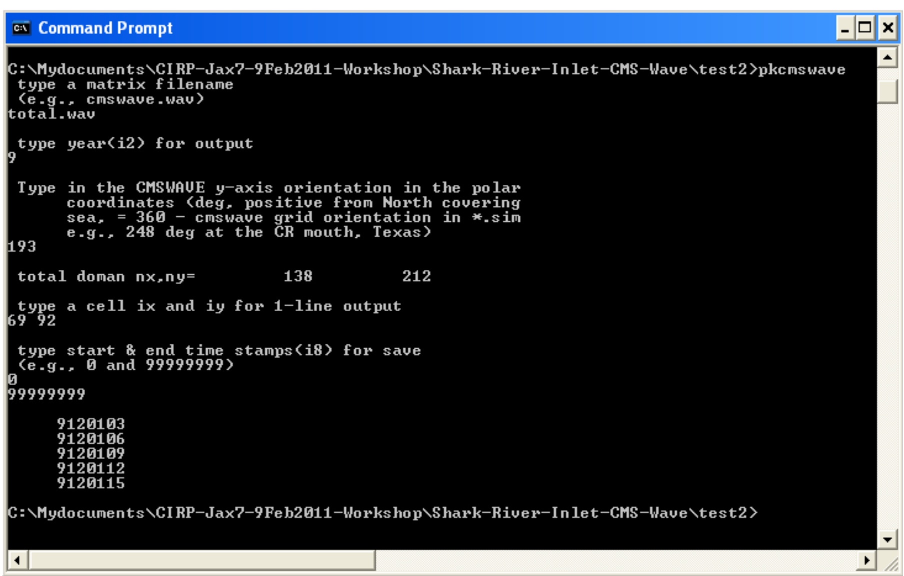


Figure 8.2.2 Run pkcmswave.exe.

- **pkcmswave1.exe –** this code will extract the calculated wave height, period and direction information from the monitoring station output file selhts.out (exists only if the monitoring station data is specified in the *.std) for a single station. The output file is 'oneline.dat' that lists the extracted wave information in the WIS short format. The selhts.out file has a 15-column format:

  Column 1 - spectrum label (or timestamp)
  2 - i index in Cartesian (i,j) or ij index in Quad-tree mesh
  3 - j index in Cartesian (i,j) or 0 (dummy) in Quad-tree mesh
  4 - significant wave height (m)
  5 - spectral peak wave period (sec)
  6 - mean wave direction (deg, local coordinate system)
  7 - swell height (m)
  8 - swell wave period (sec)
  9 - swell direction (deg)
  10 - local-generated wave height (m)
  11 - local-generated wave period (sec)
  12 - local-generated wave direction (deg)
  13 - wave breaking index (non-zero for breaking)
  14 - max water elevation mark (m), include wave run-up if calculated
  15 - flow discharge rate (m*m/sec) at structure cells if calculated

  Note: Specify IWET = -1 or -3 to save swell and local sea data in selhts.out (swell is shown in Columns 7 to 9, and local sea in Columns 10 to 12)

  1. In the DOS window, run **pkcmswave1.exe**

  2. Responding to the on-screen input, type the CMS-Wave monitoring station output filename (e.g., selhts.out)

  3. Type the year (last two digits of a calendar year) for the data, e.g. type '11' for 2011

  4. Type the local shoreline orientation (the CMS-Wave grid y axis) in clockwise polar coordinates (deg, positive from North covering the sea, e.g., 180 deg for St Mary's Entrance, FL/GA, or 360 deg - the wave grid orientation angle in *.sim)

  5. Type a single cell i and j indices (need to be a monitoring station cell given in *.std)

  6. Type the starting timestamp (default value is 0) and ending timestamp (default is 99999999) for saving output files – see Fig 8.2.3 to run pkcmswave1 to extract wave information for Cell (69, 92) from selhts.out.

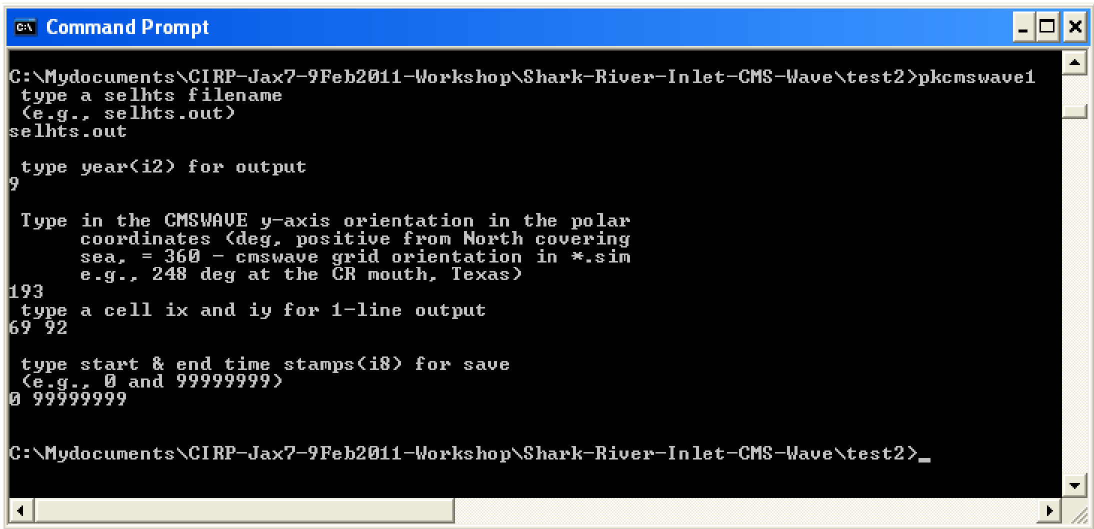


Figure 8.2.3 Run pkcmswave1.exe.

- **fig1set.m** (or **fig1set.exe**)– this MATLAB code reads a WIS short format file and plots the time series of wind and wave parameter data.

    1. In the MATLAB console, go to the folder with the fig1set.m code, type ‘fig1set’ and press *Enter* to run (or in the DOS folder, run **fig1set.exe**)

    2. Responding to the on-screen input window, browse and open (read) the WIS sort format file of interest (e.g., 4602509d.dat)

    3. Enter 1 for plots shown in the Metric system (m, m/sec) or 2 in English system (ft, knot)

    4. Type the year (last two digits of a calendar year) for the data, e.g. 11 for 2011

    5. Type starting month id number in the plot, e.g. 1 for January, 2 for February, etc.

    6. Type ending month id in the plot, e.g. 3 for March, 13 for January of the following year, or enter 0 to input starting and ending dates in the same month

    7. Type starting and ending days to display the data plot or type “0 0” and *Enter* to skip this option

    8. Type the title (text) for the plot or simply press *Enter* to skip this option, to complete the run – see Fig 8.2.4 to run **fig1set.m** or **fig1set.exe** to plot NDBC Buoy 44025 data (2009-ndbc.out) for December 2009.

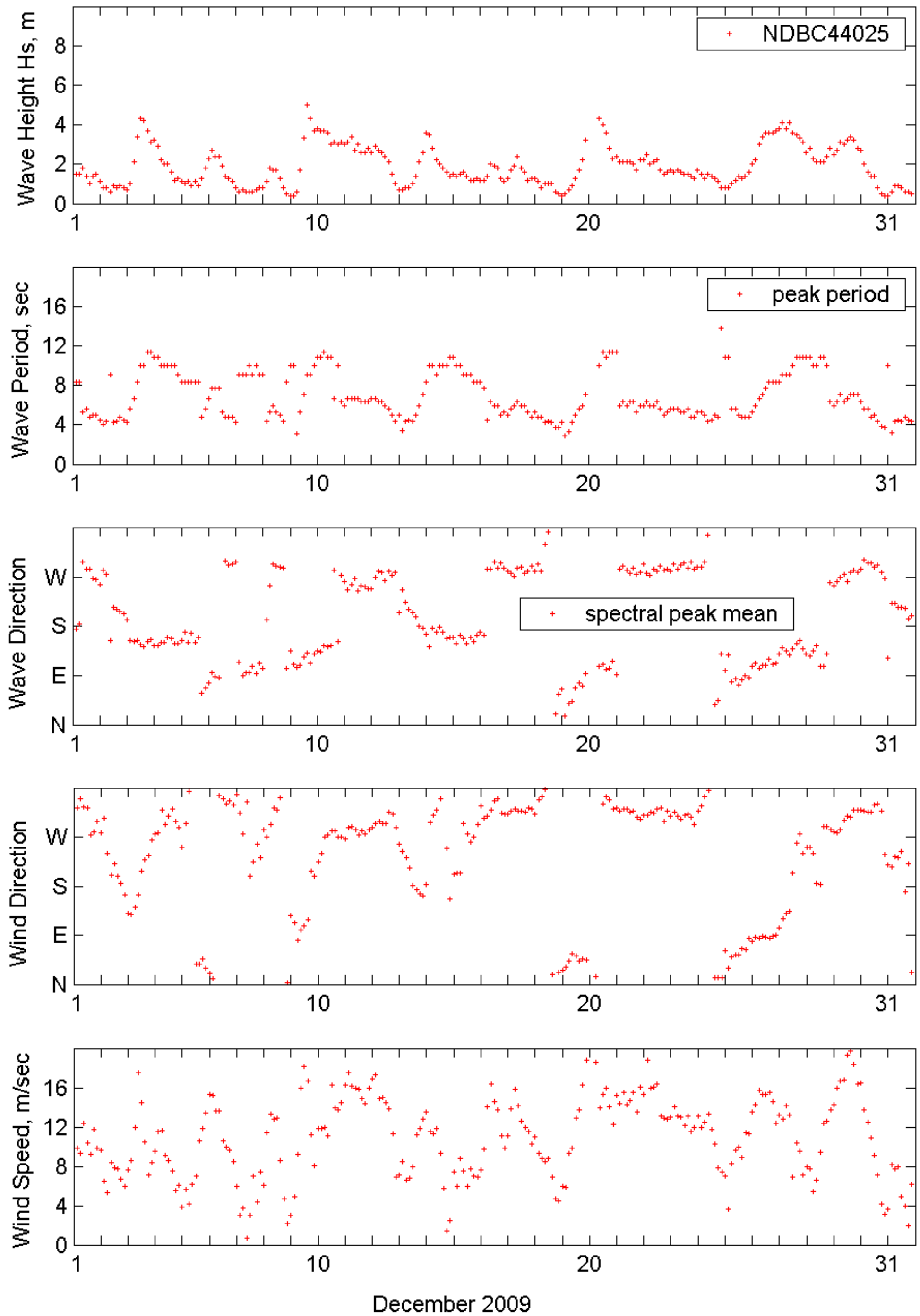


Figure 8.2.4 Run fig1set.m.

- **fig2sets.m** in MATLAB (or **fig2sets.exe** in the DOS command window) – this code is similar to fig1set.m except it is capable of reading and plotting the time series of wind wave data from two or three WIS short format files. Run **fig2sets.m** following similar steps as run **fig1set.m** – see Fig 8.2.5 to plot NDBC 44025 and CDIP 154 (2009-sp154.out) for December 2009.

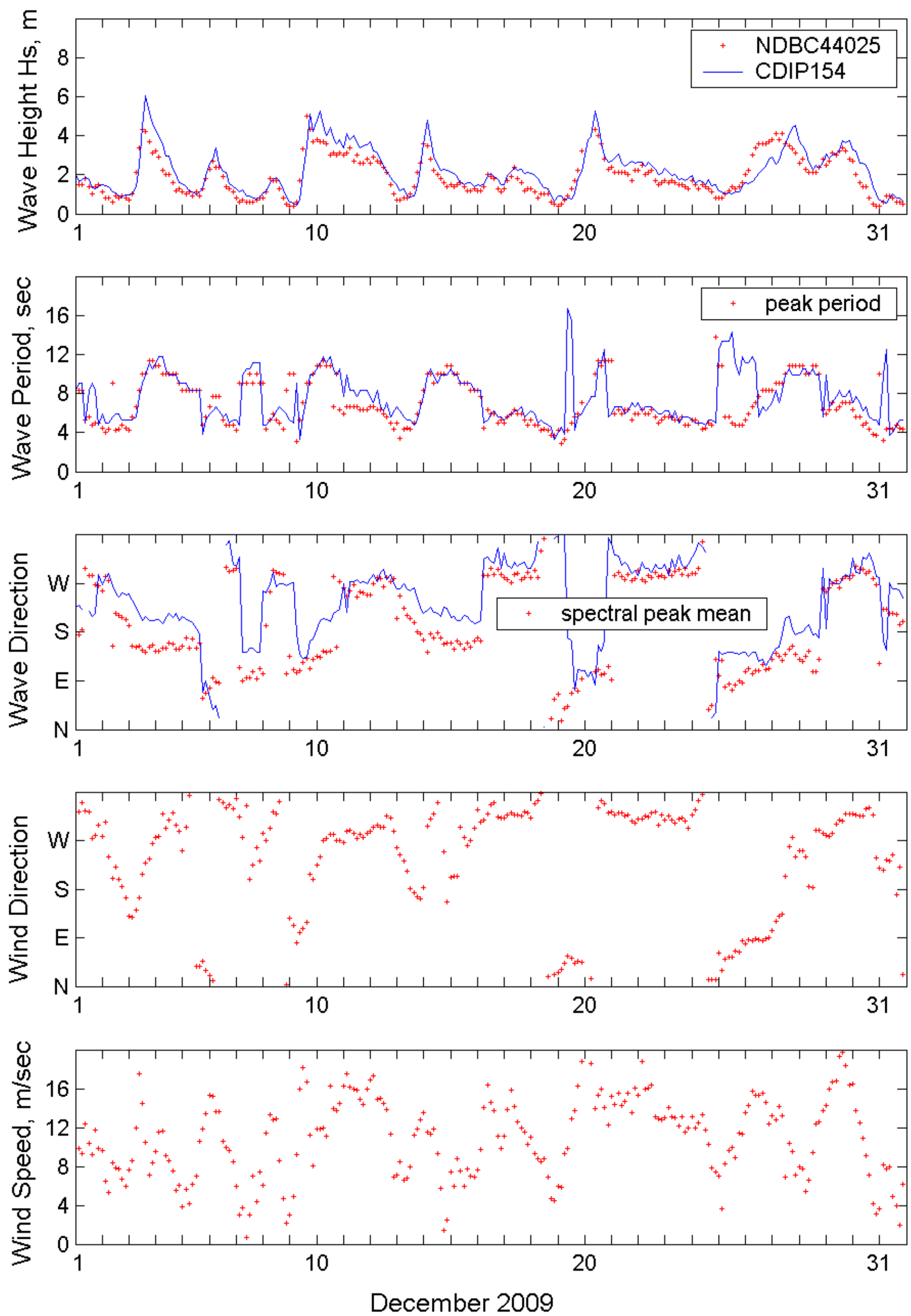


Figure 8.2.5 Run fig2sets.m.

- **fig4sets.m** (or **fig4sets.exe** in DOS) – this code is similar to **fig1set.m** and **fig2sets.m** except it is capable of reading and plotting the time series of wind wave data from three or four WIS short format files. Run **fig4sets.m** following similar steps similar as run **fig1set.m** or **fig2sets.m** – see Fig 8.2.6 to plot NDBC 44025, CDIP 154, and these two sets of data transformed to the CMS-Wave grid offshore boundary (2009-ndbc.dat and 2009-sp154.dat).

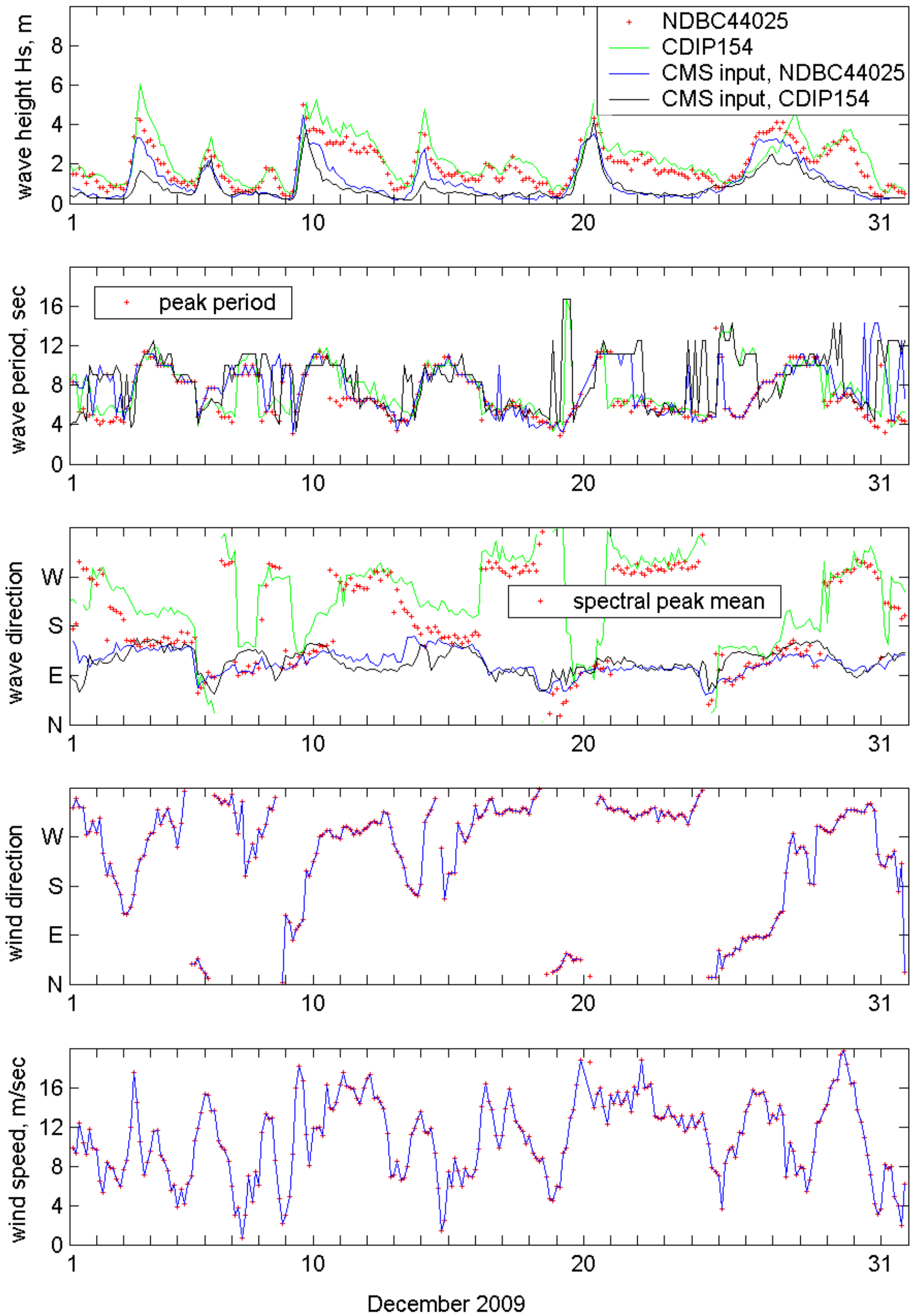


Figure 8.2.6 Run fig4sets.m.

8.3 Sediment Transport

Calculating accurate sediment transport rates is necessary for understanding trends in channel infilling, inlet and nearshore morphology change, and on a more basic level sediment pathways and checking sediment balance.

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 m3/m.

….channel infilling, nearshore sed tran, sediment budget through a channel, boundary fluxes.

- Calculating sediment fluxes over cross-sectional arcs:

    1. Load the *Jan09Tel.cmcards* into SMS - the solution file should load in automatically

    2. Right-click on the default coverage and select *Type | Generic | Observation* to set up the map module for drawing an observational arc (Figure 8.3.1)

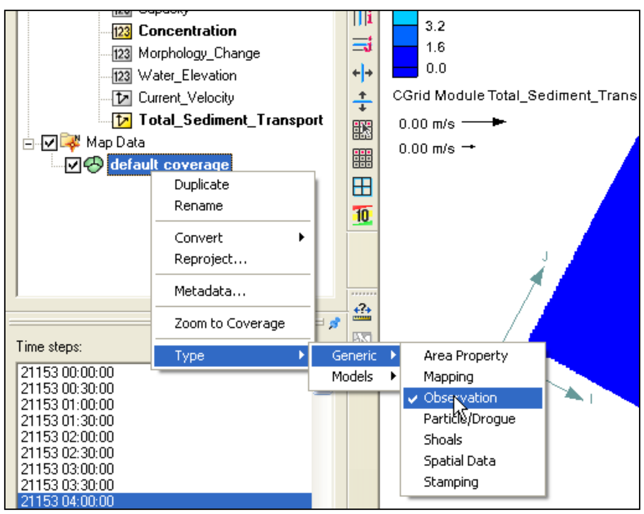


Figure 8.3.1. Changing a *Map coverage* type to *Observation.*

    3. Zoom to the location selected for calculating sediment transport (Example of a cross-shore location shown in Figure 8.3.2) 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

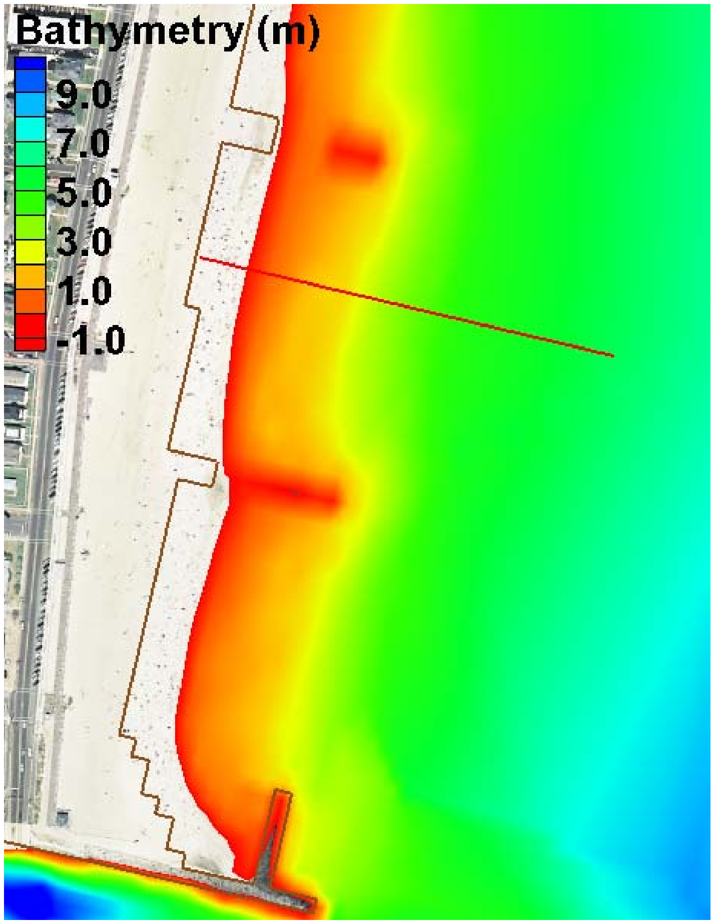


Figure 8.3.2. Example of a *Feature Arc* for calculating sediment transport.

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 8.3.3. 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.

4. 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  and right-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.

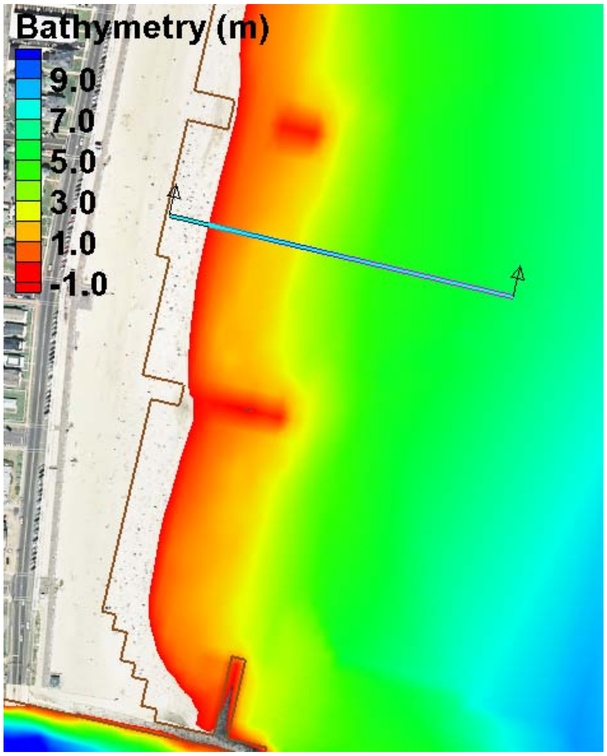


Figure 8.3.3. Highlighted Feature Arc with the positive direction shown by the arrows.

5. 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 same applies for feature polygons.

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 Cartesian Grid Module.

6. To create this dataset, select the Cartesian Grid Module, Click on *Data, Data Calculator*

7. In the *Data Calculator* (Figure 8.3.4), 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.

8. 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.

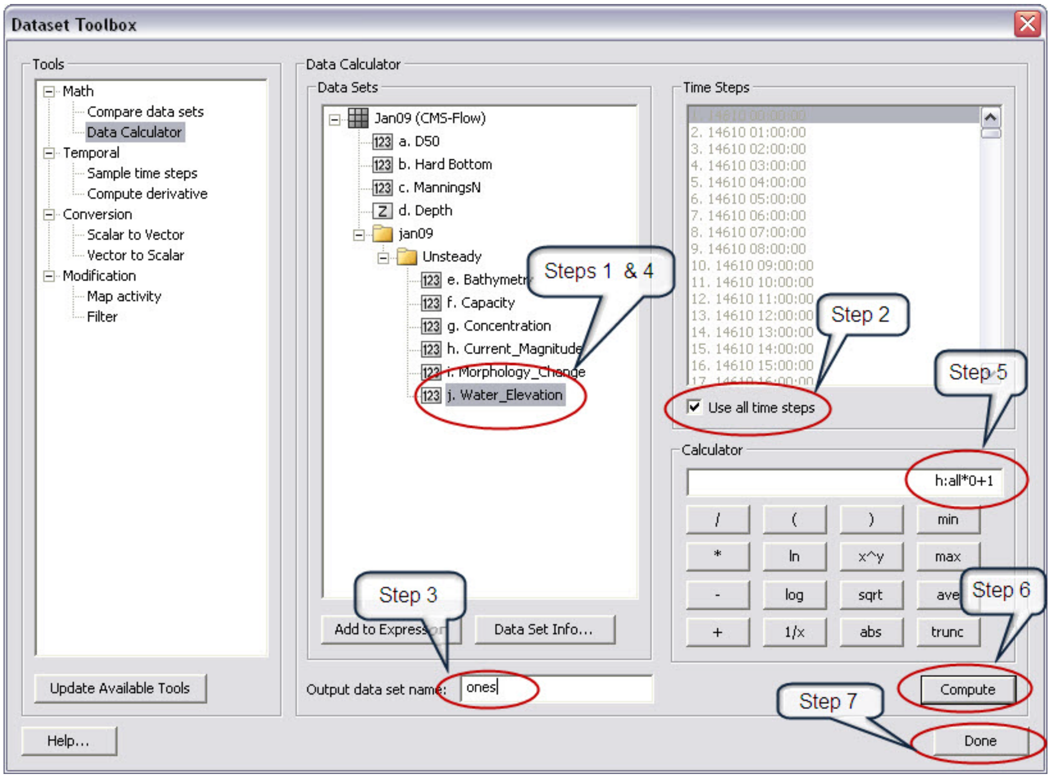


Figure 8.3.4. Steps to create a time series dataset of "ones" within the Cartesian grid module.

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:

9. 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*.

10. Check the feature arc checkbox (Figure 8.3.5), 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.)

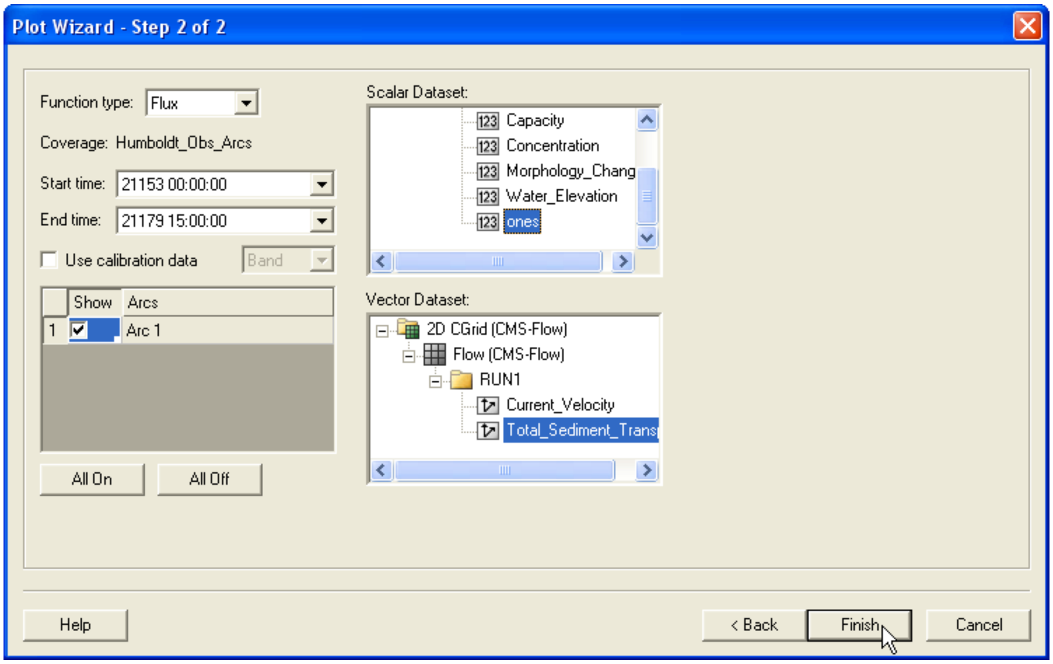


Figure 8.3.5. Selecting the correct *Start* and *End Times*, *Feature Arcs*, and *Scalar* and *Vector Datasets* in Step 2 of 2 of the *Plot Wizard.*

Once the time series dataset is created in the form of the graph (Figure 8.3.6), the data must be exported for calculations in other software as SMS does not yet have the capability to do so. 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

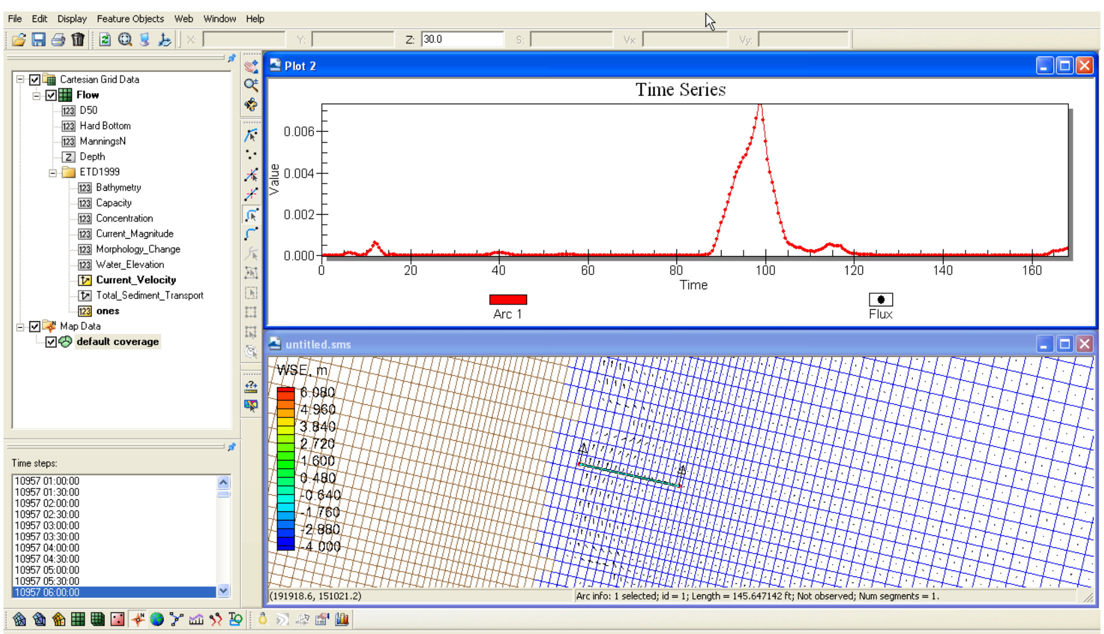


Figure 8.3.6. Example time series of integrated sediment transport across a *Feature Arc*.

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.

11. From the generated graph, export the text data by rt-clicking on the time series plot and selecting *View value*

12. From the *View Values* window, select the data from the table (Figure 8.3.7) and copy-paste it to a text editor such as Wordpad. (Note the number of rows in the dataset by looking at the id number of the last row in the *View Values* window.)

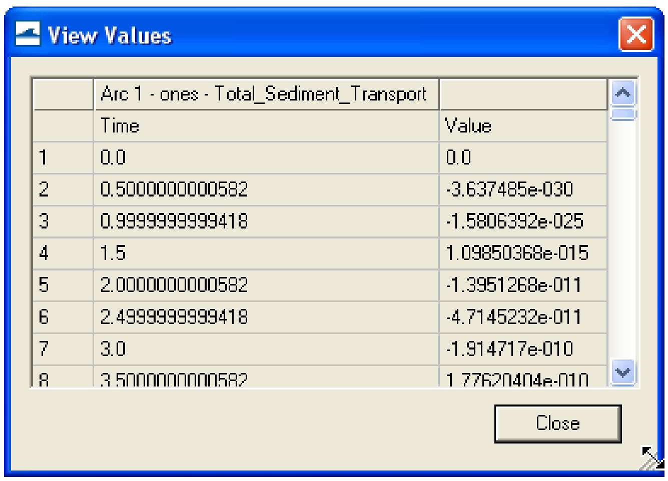


Figure 8.3.7. Example of a *View Values* window in SMS.

13. Save the file with the extension **.txt* (Figure 8.3.8). The data may also be saved as an *.xys file (SMS xy series file), where the file needs a header (as shown in Figure 8.3.8). 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. (An example is given in the folder for Section 8.)

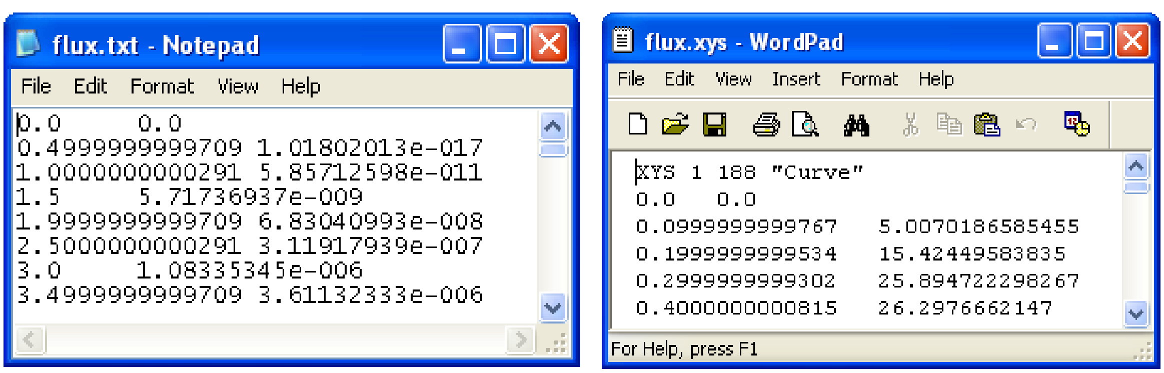


Figure 8.3.8. Examples of *.txt (left) or *.xys (right) files with extracted sediment transport time series.

14. Double click on the executable named *proc_flux.exe* (Figure 8.3.9). 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.

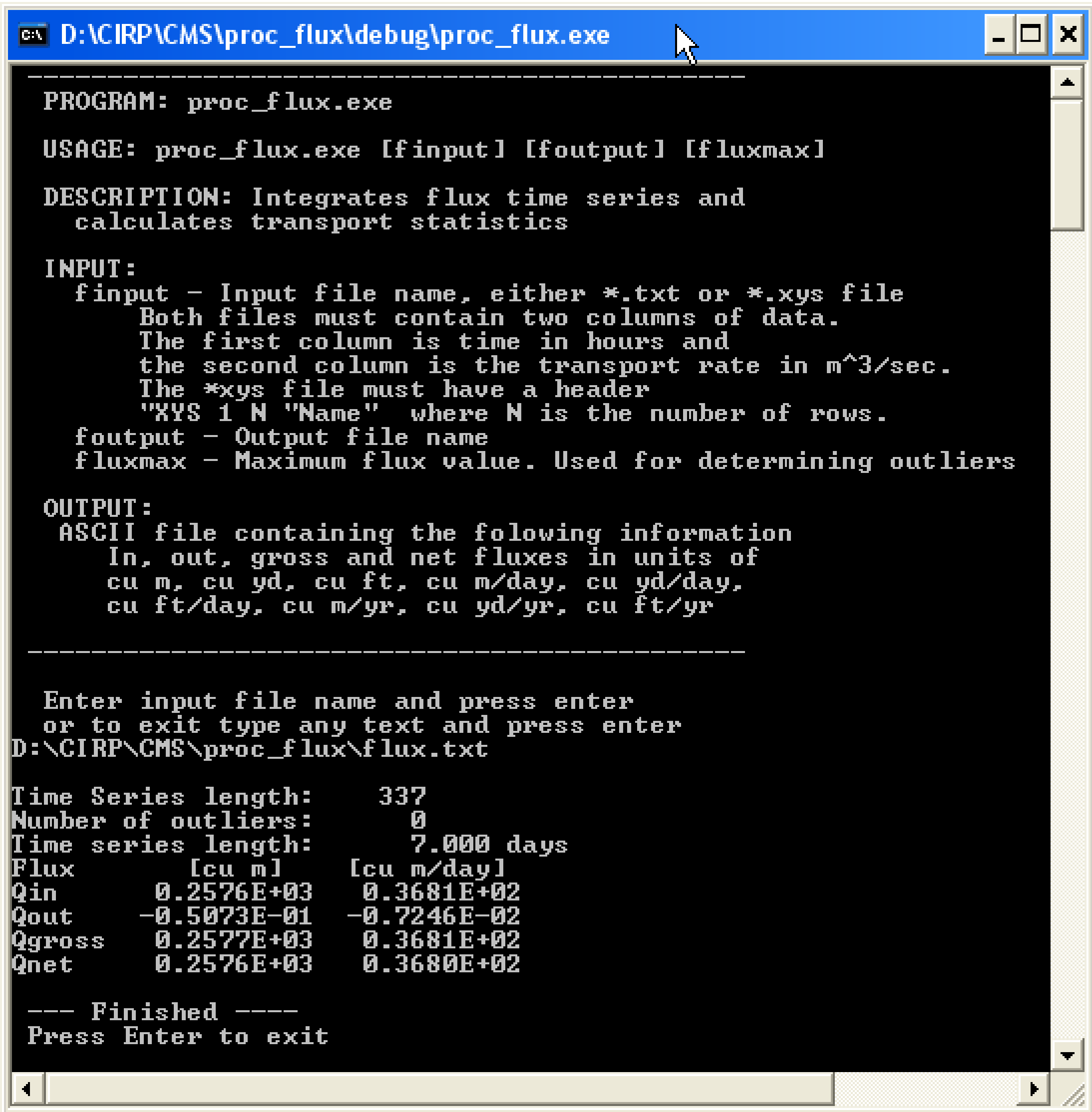
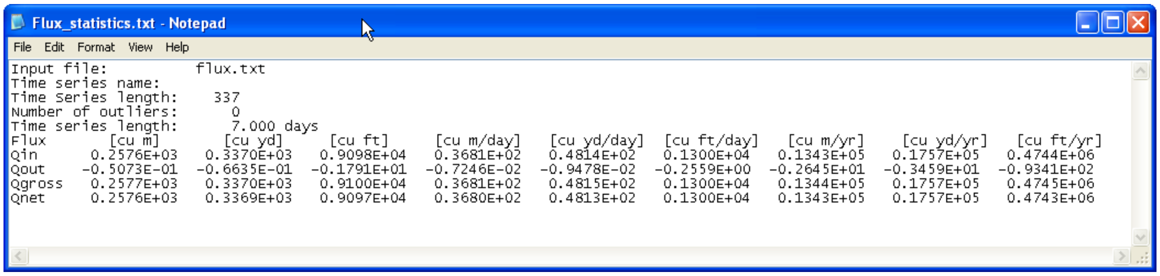


Figure 8.3.9. Example calculation of sediment transport statistics using the proc_flux.exe from a command prompt. Note the input file name contains the directory location of the text file.

15. 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 8.3.10.




```
Input file:            flux.txt
Time series name:
Time Series length:      337
Number of outliers:        0
Time series length:        7.000 days
Flux        [cu m]       [cu yd]      [cu ft]    [cu m/day]   [cu yd/day]   [cu ft/day]    [cu m/yr]   [cu yd/yr]   [cu ft/yr]
Qin      0.2576E+03   0.3370E+03   0.9098E+04   0.3681E+02   0.4814E+02   0.1300E+04   0.1343E+05   0.1757E+05   0.4744E+06
Qout    -0.5073E-01  -0.6635E-01  -0.1791E+01  -0.7246E-02  -0.9478E-02  -0.2559E+00  -0.2645E+01  -0.3459E+01  -0.9341E+02
Qgross   0.2577E+03   0.3370E+03   0.9100E+04   0.3681E+02   0.4815E+02   0.1300E+04   0.1344E+05   0.1757E+05   0.4745E+06
Qnet     0.2576E+03   0.3369E+03   0.9097E+04   0.3680E+02   0.4813E+02   0.1300E+04   0.1343E+05   0.1757E+05   0.4743E+06
```

Figure 8.3.10. 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. Below is an example of calculating the sediment fluxes through the inlet ebb shoal at Shark River Inlet.

- Arc box around the inlet shoals to capture sediment balance

  1. As instructed in the above section describing calculating sediment fluxes across arcs, here sediment flux transects were chosen along assumed boundaries of the inlet's shoal.

  2. Create arcs using the *Create Feature Arc* tool , similarly to Figure 8.3.3, by making each quantifiable transect (arc) individually, and by starting the next arc on the end *Feature Point* of the last arc. This ensures that the arcs close the loop.

  3. After the loop of arcs is closed, go to *Feature Objects*, and *Build Polygons* should be visible to select. If *Build Polygons* is grayed out, then the arc lines were not closed. To fix this, zoom in on the connecting Feature Points to determine which are offset. Move the points closer, highlight them, and select *Feature Objects, Clean*, and check *Snap Nodes and Vertices*.

  4. Sediment transport fluxes can be obtained by following Steps 9 through 15 of the above section (after creating the "ones" time series dataset) for each individual arc.

8. Post-processing Results

8.4 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

    1. Load *Jan09tel.cmcards* (and solution file, if it does not load automatically) located in the directory *Section 8.4* directory

    2. Rt-click on the *Time Series* window on the left and select “Time Settings”, and change the time reference from *Relative* to *Absolute*

    3. In the Cartesian Grid Module, click *Data*, *Data Calculator*, and the Data Calculator should be selected in the Tools section as shown in Figure 8.4.1

    4. Select the *Bathymetry* scalar ( ) dataset; under *Time Steps* check the *Use all time steps* on, and double-click the *Bathymetry* (highlighted in Figure 8.4.1)

    5. The line under Calculator will display “e:all”, add the multiplication symbol (*) and negative 1 (-1) as shown in Figure 8.4.1

    6. Change the *Output data set name* as shown in Figure 8.4.1 so as to distinguish this dataset from the original *Bathymetry* output, click *Compute*, and after the scalar set appears in the SMS window (Figure 8.4.2), click *Done*. (Note: values for the contours may need to be adjusted to view elevation range.)

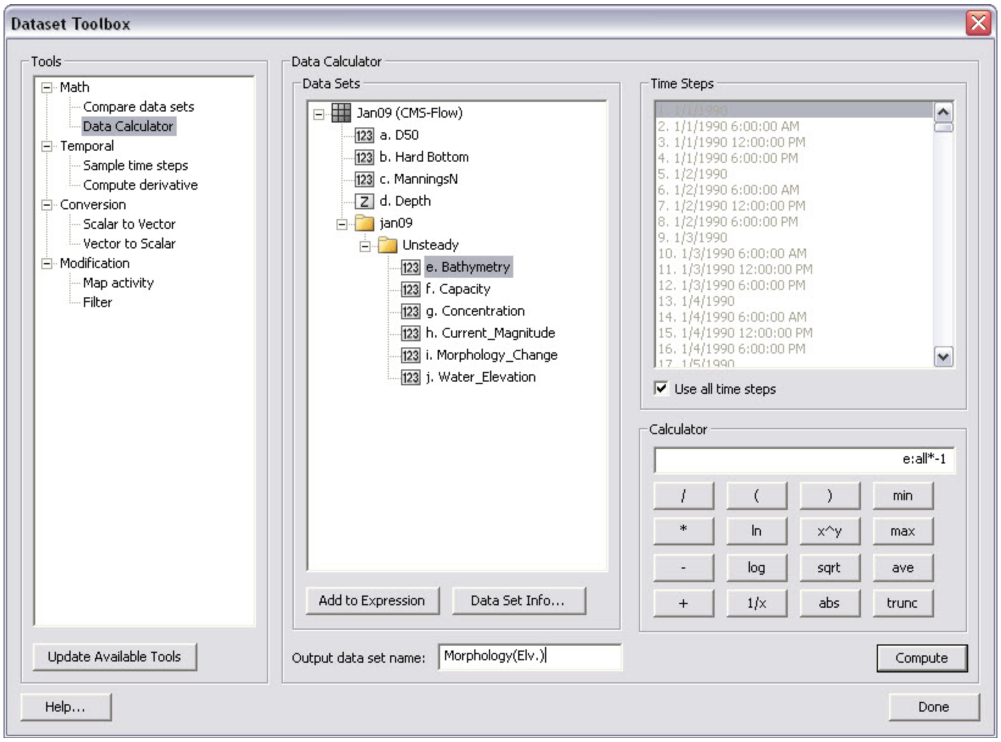


Figure 8.4.1. ***Data Calculator Options*** **in the** ***Dataset Toolbox*****.**

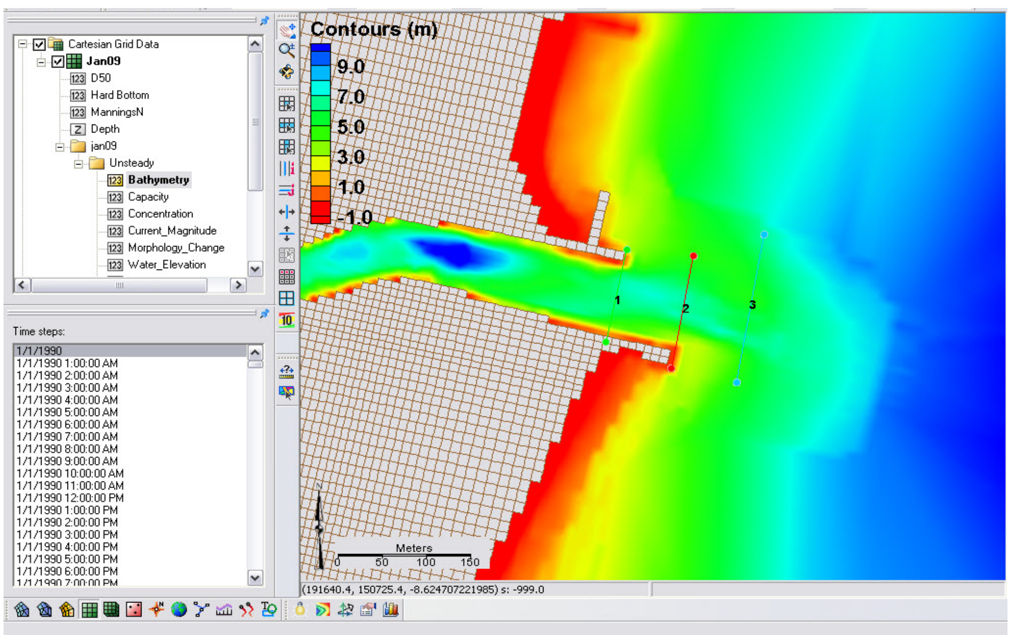
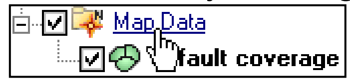


Figure 8.4.2. Display of calculated bathymetry now converted to elevations. Note the legend has been modified to display the new contours.

7. Create feature arcs for extracting the time series data by switching to the *Map Module* button , or Map Data 

, rt-click on *default coverage*, and select *Type*, *Generic*, *Observation*

8. Click the *Create a Feature Arc* button  to generate the 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 the arc

9. 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)

10. Rt-click an arc and click on *Attributes*, and set the color and name(s) of each arc which are listed together in the *Observation Coverage* options (Figure 8.4.3), click *OK*

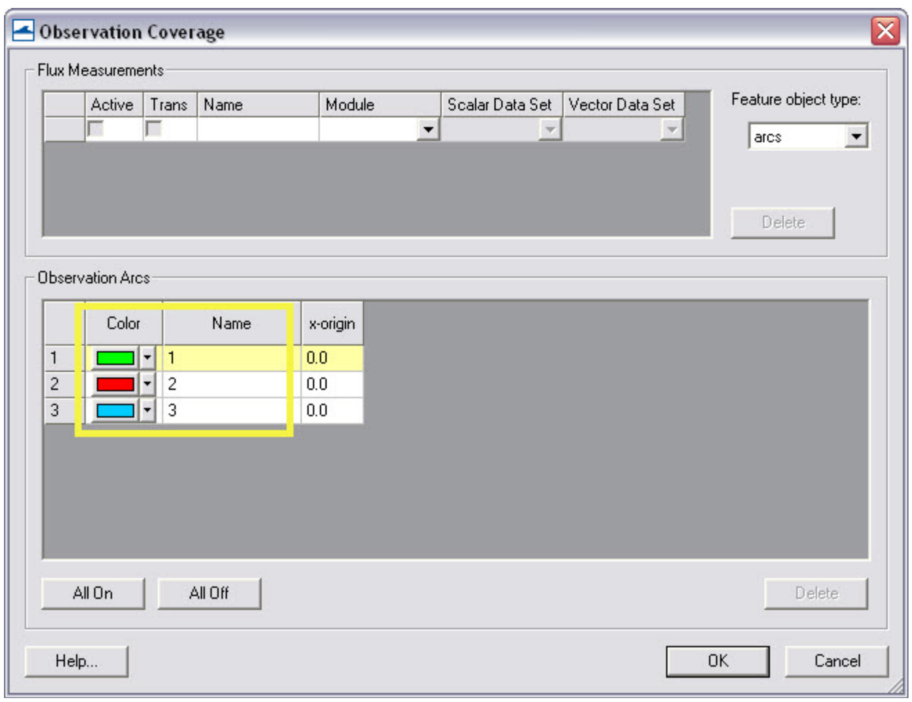


Figure 8.4.3. ***Observation Coverage*** **attributes.**

11. To plot the profiles in SMS, Select *Display*, *Plot Wizard*, and *Observation Profile*, and select *Next*

12. Following Figure 8.4.4, under *Coverage* check one arc (though multiple can be plotted), under *Data set* select *Specified data set(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*

13. Rt-click on the plot (Figure 8.4.5) 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)

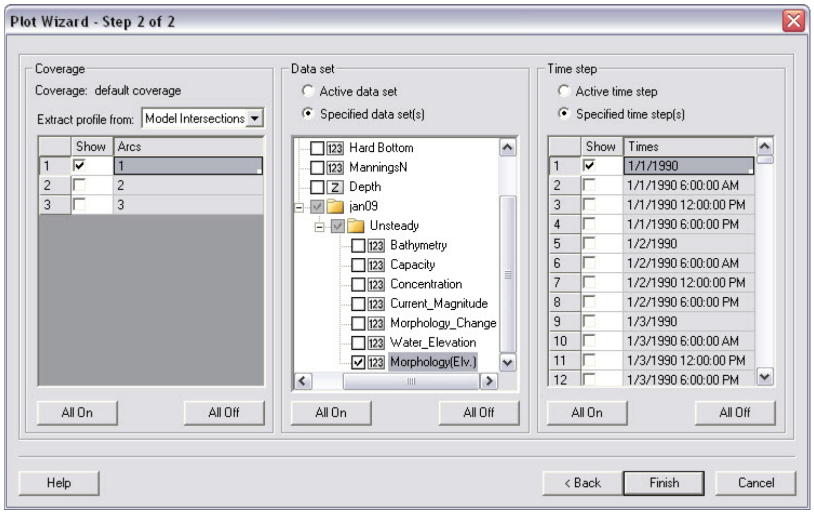


Figure 8.4.4. *Plot Wizard* Step 2.

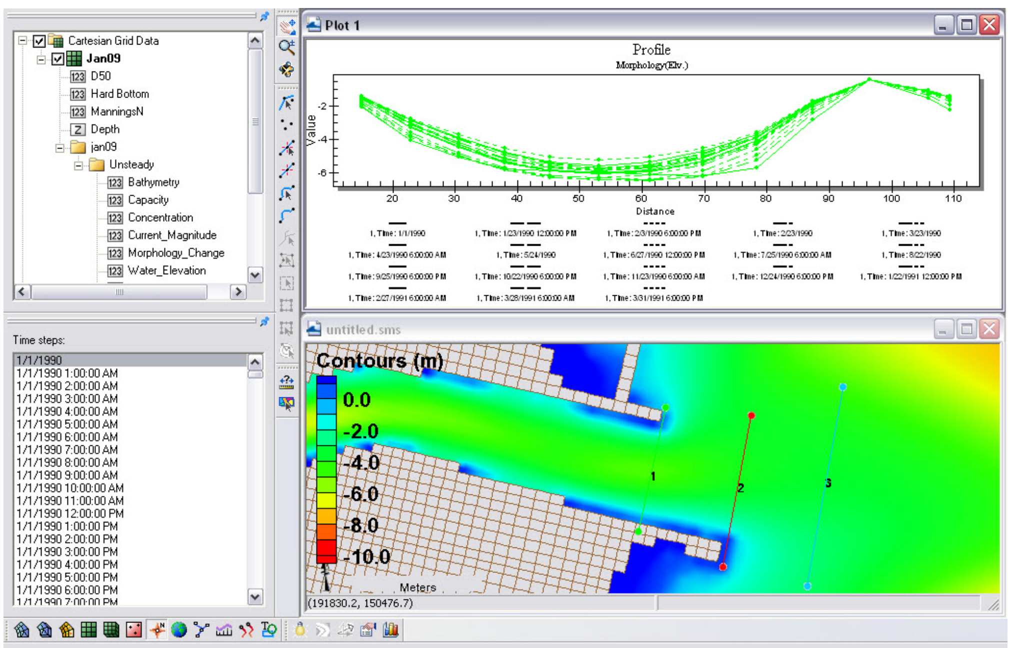


Figure 8.4.5. Plotted cross-section.

14. To export the data in different formats, rt-click on the plot, and select *Export/Print* for several options to export the graph or data from the graph (Figure 8.4.6)

15. To add more profiles or time-series, rt-click on the plot, and select *Plot data..* to bring up the original options from Figure 8.4.4.

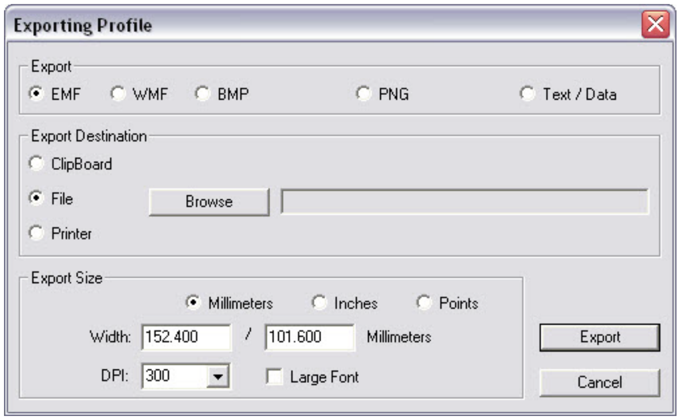


Figure 8.4.6. *Data Export* Options.

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 off of 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 off of this analysis alone.

- Planform area changes

    1. To filter the morphology dataset for the time periods of interest, i.e. every month, open the *Data Calculator* and select *Sample time steps* under the *Temporal* section (Figure 8.4.7).

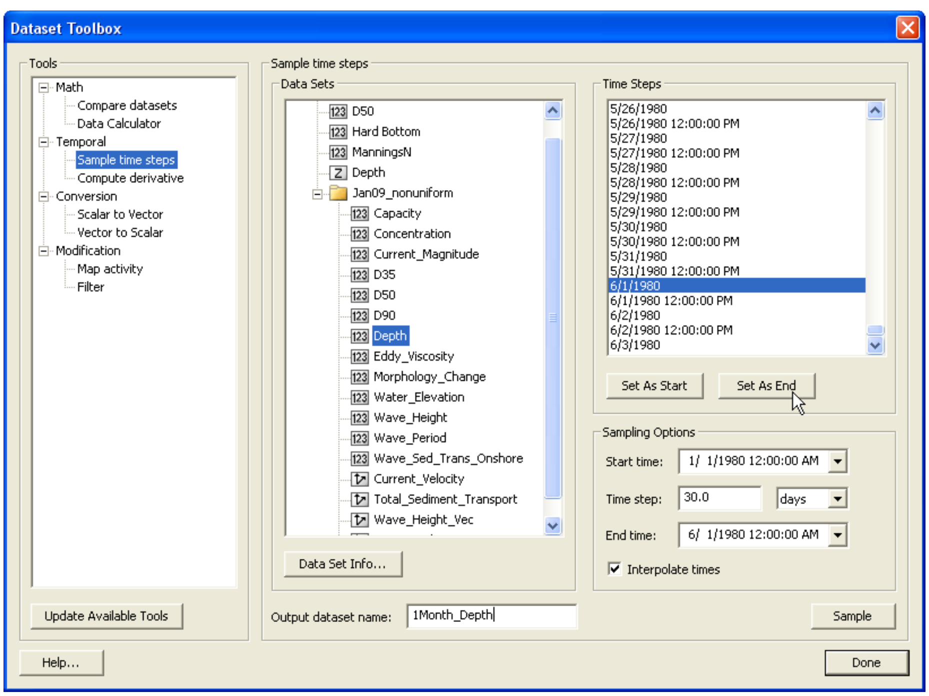


Figure 8.4.7. Sampling time steps from a solution dataset.

2. Select *Depth* under *Data Sets*, change the *Output dataset name*, and set the *start time* and *end time* to Jan 1 and Jun 1, respectively, with an interval of 30 days. Select *Sample*. This should produce a *Depth* timestep approximately every month.

3. In order to map an elevation, the grid must be converted to a scatterset.

   - Rather than take the time required to convert the entire solution during this workshop, we have provided this file in the folder (File: *1Month_Depth*). (To convert the grid solution file to a scatterset, select *Data|Map to Scatter*, and SMS will convert all of your time series datasets to the scatter module (Figure 8.4.8). This can take anywhere from minutes to hours, depending on the size of the file and speed of the processor. Open the provided scatterset.

   - Or, delete the Jan09_nonuniform solution datatree so that the main CMS-Flow datasets and the *1Month_Depth* dataset are the only datasets left. Then select *Data|Grid-> Scatter*, and SMS will convert only those datasets.

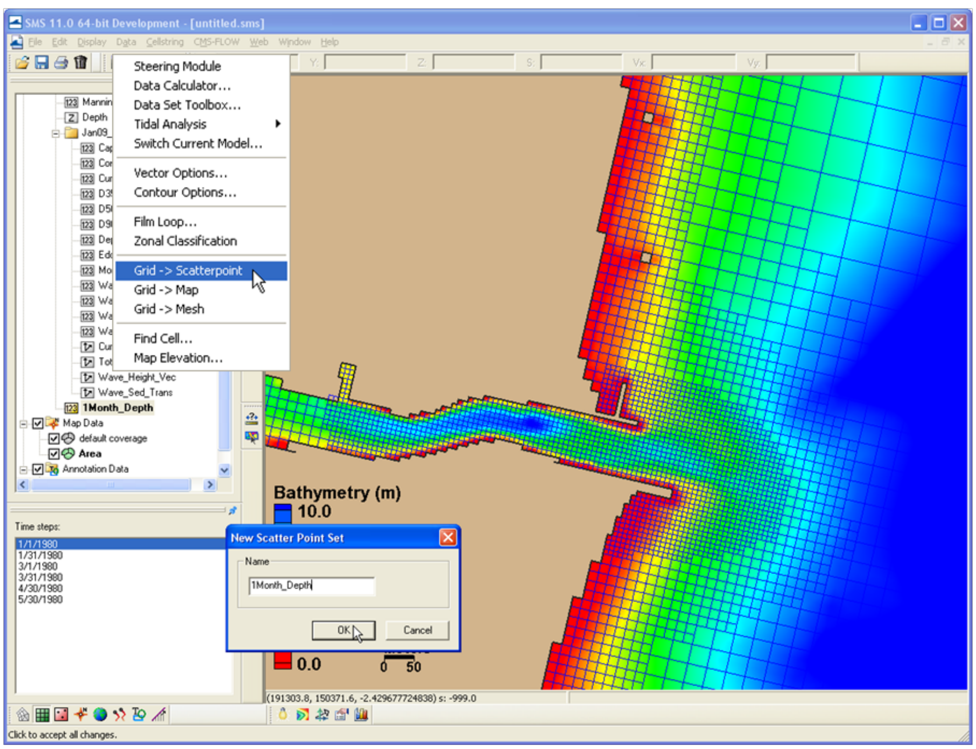


Figure 8.4.8. Converting a solution dataset to a scatter dataset.

4. Create a new map coverage (by right clicking *Map Data* in the data tree and selecting *New Coverage*) for mapping out specified contours over the time series depth scatterset.

5. Select the *1Month_Depth* Scatterset (*Scatter Module* should be on) and the first time stamp (1 Jan), and go to *Data|Scatter Contour -> Feature.* Choose an elevation to map (6.0 m), a spacing of the points in the resultant arc, and select the Map Coverage ('*Area*' in Figure8.4.9) that the arcs are created in.

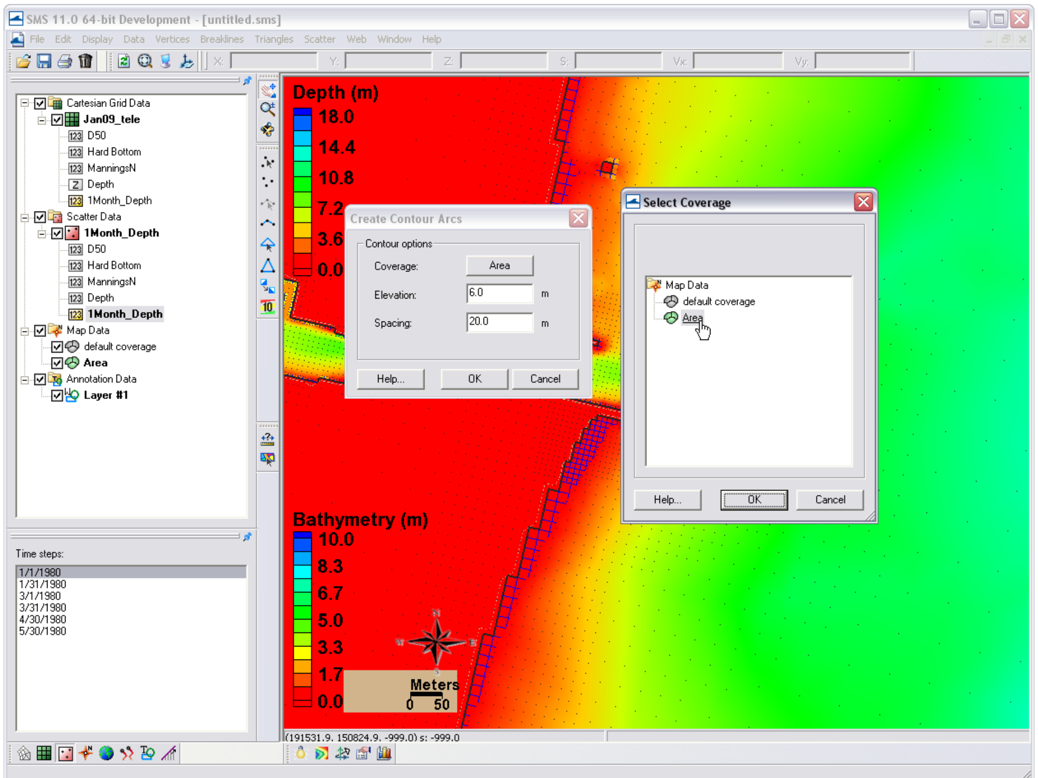


Figure 8.4.9. Mapping a scatterset contour to the map module.

6. Create another new Map Coverage, select the final timestamp (30 May), and go to *Data|Scatter Contour -> Feature* and select the same elevation (6.0 m) and point spacing. Switch between the two coverages to view the location of each contour (Figure 8.4.10). Note that the arcs hold a z-elevation (displayed in Z in the bar above) and are partially visible under the color-filled scatter data.

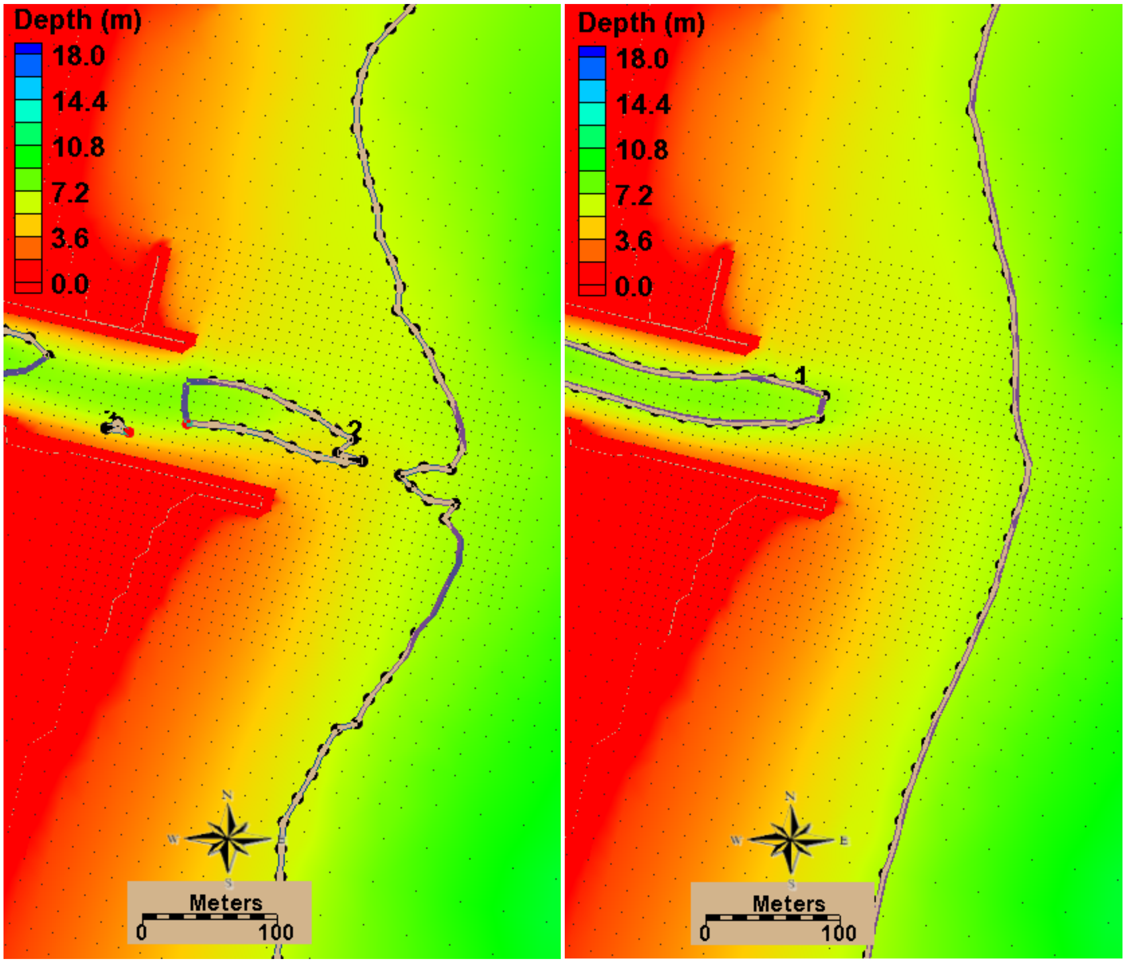


Figure 8.4.10. Mapped coverage of the 6-m contour at Months 1 and 6 (left and right, respectively).

7. To calculate planform area, each contour in both maps must be closed off with arcs to create a polygon. Create a polygon in the closed arcs by selecting *Feature Objects|Build Polygons*. An example is shown in Figure 8.4.11. Note that the area of the selected polygon is given at the bottom of the screen in the units the solution file is given in, which is always metric for the CMS.

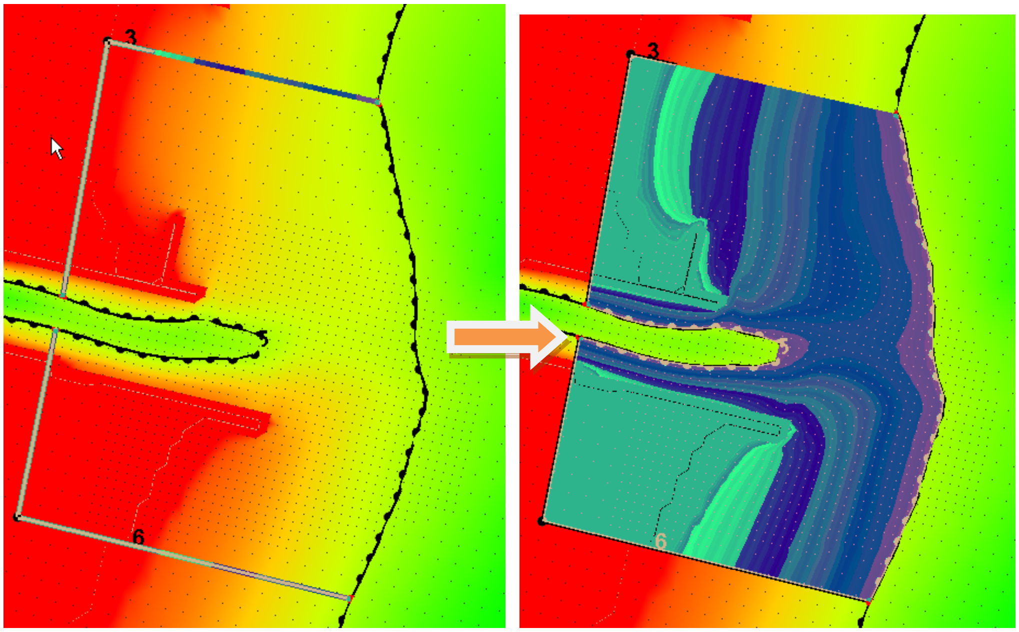


Figure 8.4.11. Left: Two arcs drawn to connect the 6m contour back to a common point to close off the polygon. Right: Highlighted polygon after created.

Creating polygons between time series allows for calculation of planform area change at a morphologic spatial scale through time. The difference shown in the arcs in Figure 8.4.10 is substantial, and can be quantified with the use of arcs and polygons. It is important to be careful with delineating the morphologic form of interest. If the full area is being calculated for a feature (an ebb shoal), be sure to be discrete in the size of these aerial calculations because they may skew the final calculation of the morphologic feature of interest.

Similarly, volume change can be elicited from polygons created with Feature Arcs. The next sections describes how to calculate volume change over an arc.

- Calculating volume change from polygons

    1. Create a polygon similarly to the above exercise that created arcs to complete a polygon in the planform area calculations. Choose an area around the ebb shoal, or within the channel, and draw a single arc connecting around the feature. Be sure to close the end of the polygon by letting the mouse automatically hover over the final point (the pointer will relocate when placed near another feature point).

    2. Create a polygon in the closed arcs by selecting *Feature Objects|Build Polygons*. An example is shown in Figure 8.4.11.

    3. To select the cells within the polygon, next, with the polygon selected, go to *Feature Objects|Select/Delete Data*, and select *select*, *Cartisian Grid Cells*, and *Inside Polygon* (Figure 8.4.12).

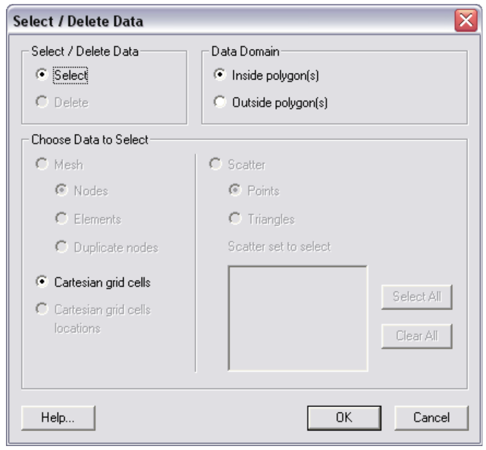


Figure 8.4.12. Options for selecting and deleting data from Mesh, Cartesian Grids, and Scattersets using polygons.

Note, again, that the volume of the selected polygon is given at the bottom of the screen in the units the solution file is given in, which is always metric (m^3) for the CMS. The volume given in the lower right corner of the window (Figure 8.3.13) are a positive, negative, and total volume representative of the integrated positive, negative, and combined volumes from each cell. Therefore, the numbers represent the volume based off of the dataset's datum.

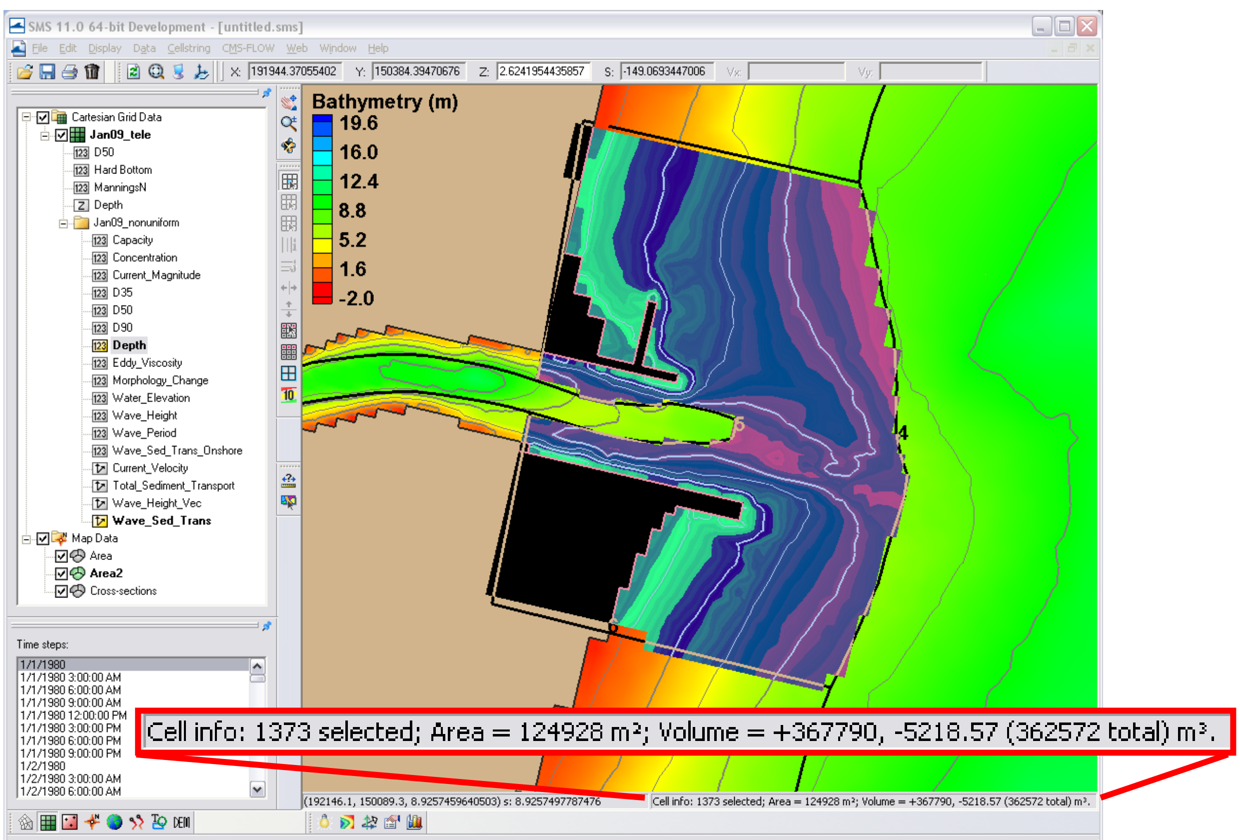


Figure 8.4.13. Selected cells from a polygon. Volume information given in lower, right corner of screen.

If depths are selected in the solution datatree as the measurement for volume calculation (one of the direct outputs in the CMS solution file), then volume is positive and represents the space that the water occupies. To get the volume of a morphologic feature, a base level to calculate above must be chosen. Navigation depths or depth of closures are often used as a base level for a quick estimation of volume. The area of the cells is also included in basic cell info at the bottom of the screen, as illustrated in Figure 8.4.13. Multiplying this area by the base level (with respect to the model datum) will produce a base volume of which the calculated volumes can be subtracted from.