US Army Corps of Engineers
BUILDING STRONG®

Coastal Inlets Research Program
11th Annual Technology-Transfer Workshop

Hands-On Tutorial

Humboldt Bay, CA

U.S. Army Engineer Research and Development Center
2 Dec 2009

Humboldt Bay, CA

Topics:

14.a. Grid generation and Preconditioning
14.b. CMS-Flow setup
    1. Hydrodynamics
    2. Sediment transport and Morphology change
    3. Salinity
14.c. CMS-Wave setup
    1. Grid Transformation
    2. Input Spectra
    3. Model Parameters
    4. Runup and Overtopping
    5. Structure Types
14.d. Calculating channel infilling
14.e. Calculating sediment transport paths and sediment budgets with CMS
14.f. Breaching and morphology change near jetties

14.a. CMS Grid Generation and Preconditioning Hands-On Practice: Humboldt Bay, CA

Introduction

The two fundamental steps in developing well-calibrated hydrodynamic and sediment transport model are to (1) created an accurate bathymetry and topography (initial) grid, and (2) apply the correct boundary forcing. Here, we discuss Item 1. One must represent the morphology of the entire domain well, particularly closest to the area of interest, with attention paid to vertical datums if different bathymetry data sets are being combined, which is typical. Bathymetric datasets from which the initial grid is generated should be of the highest quality available, and any necessary preconditioning or decrease in resolution can be addressed after the bathymetric dataset is developed.

Spatial datasets may include:

- Navigation channel surveys
- Nearshore elevations
  - LIDAR/SHOALS, beach profiles
  - Shoreline
- Digital Elevation Model (DEM)
- Offshore bathymetry

Bathymetric and shoreline datasets are essentially the base files. Refinements of the beach elevation provide the best representation of the nearshore region and are necessary for a more accurate longshore sediment transport calculation. At sites where the continental shelf is shallow, high resolution of offshore shoals (e.g., transverse shoals) may need to be represented. For cases of overland flooding, or barrier breaching, detailed land elevations may be necessary to capture the surge and subsequent breaching.

Import All Bathymetric and Elevation Datasets

All datasets should primarily contain XYZ (spatial and vertical location) and be in a text format. Most horizontal and vertical datums can be converted within the SMS scatterset utility, or *Scatterset Module*.

For the Humboldt Bay site, you are given the XYZ files to start with in a text format (a point file). They include the bathymetry from a sampled (filtered) LIDAR dataset, a sampled multibeam survey of the nearshore, inlet entrance, and larger channels in the back bay.

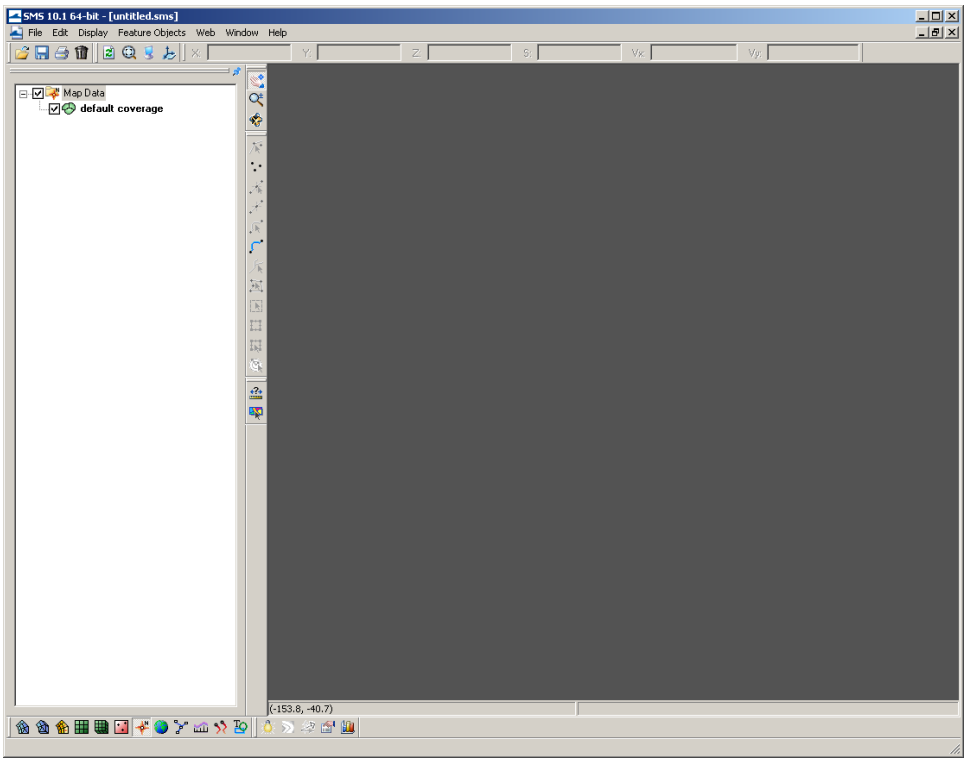


Figure 1. SMS base screen. Text files and most SMS generated files can be dragged and dropped into the grey area.

- Importing XYZ files and Setting Project Location

    1. Open SMS (Figure 1),
    2. Click on *File*, *Open*, or click the button,
    3. Go to the *14 - Grid Generation* folder, *XYZ-Initial*,
    4. Select all three datasets (), *Open*,
    5. Make sure each column is separated, click *Next*
    6. Make sure the T*ype* lines up with the X, Y, and Z columns, click *OK* to bring the files in to SMS,
    7. Click on *Edit*, *Projection* (Figure 2), and set the *Global Projection* to State Plane Coordinate System, NAD 83 *Datum*, Meters *Units*, California Zone 1 (FIPS 401) *Zone*, and the *Vertical Projection Units* to Meters, click *OK*,
    8. Save the project (as an SMS file) by clicking *File*, *Save As*, and *Save* the project.

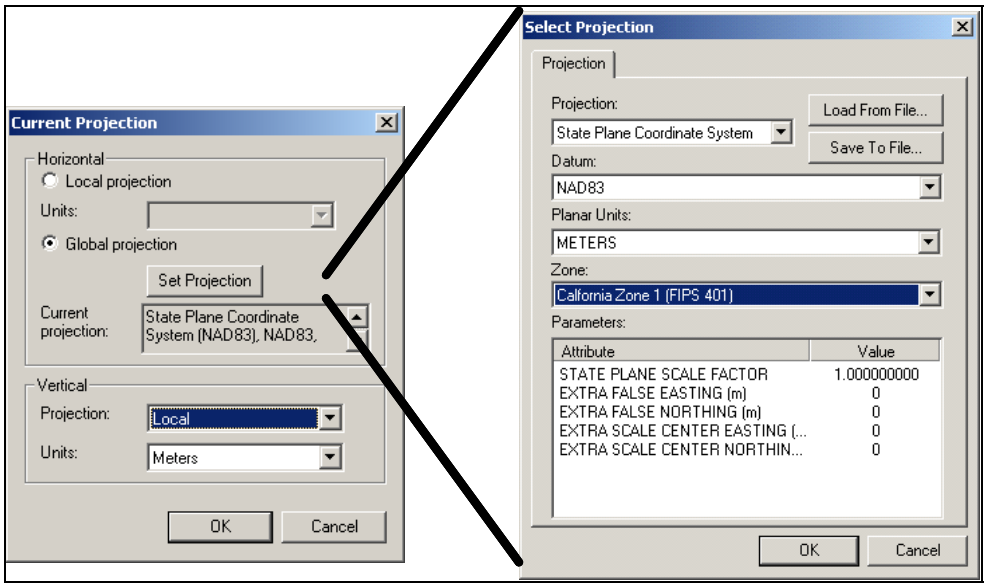


Figure 2. Project projection options.

Note: Step 6 has several options one can use to modify the files being imported. The data type is already selected as a scatter set type file, but other spatial information files can be imported this way, and selection of the type will provide more options for the columns other than x, y, and z (e.g. wind, wave, and water level files). Filtering is an option available during the import, but is also available as a data modification option in SMS. The filter option is a useful tool for importing large files such as LIDAR or multibeam bathymetry.

- Change SMS Display Options

    1. Click on *Display*, *Display Options*, or the button,

    2. Go to the *Scatter* tab in the Scatter options, click to check the points off and contours on,

    3. Go to the *Contour* tab, change contour method to *Color Fill* and adjust the data range as necessary (-5.0 to 30 m) (Figure 3).

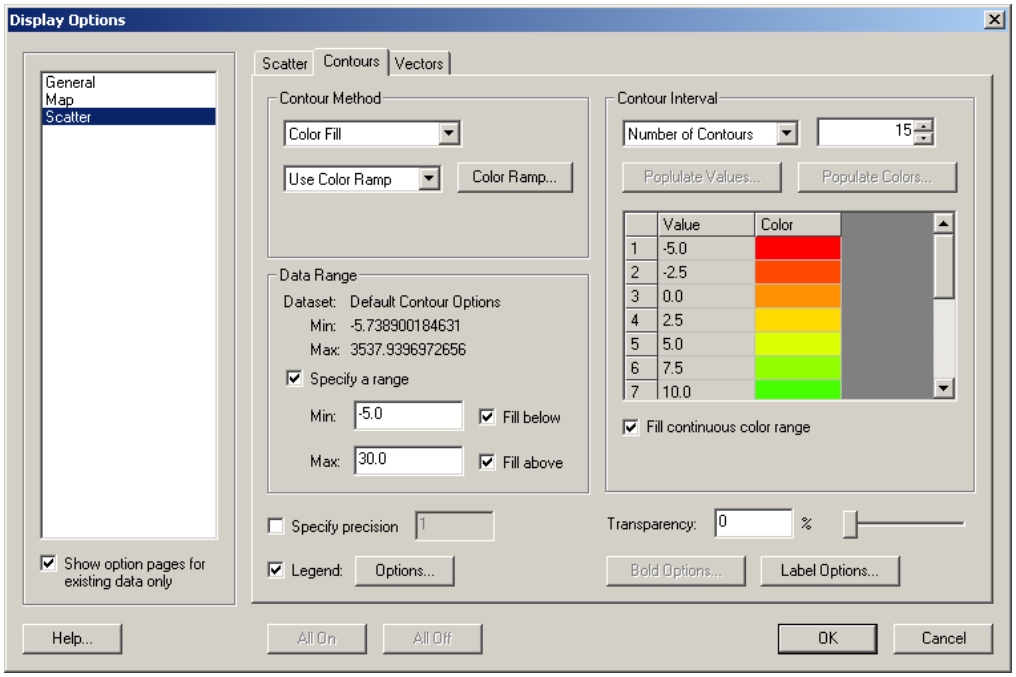


Figure 3. Display Options

Each dataset can be viewed separately by clicking on each name on the left side (the files stay separated in SMS). If the points are checked on, the selected set will display the points as the color you define in the display options and all other datasets can be seen as a faded color.

The next step will be to combine each file into one scatter set file. Rectified aerial photos are provided as a background for assisting with scatterset conditioning. For convenience, all files have been saved into the state plane coordinate system for northern California (California Zone 1 - FIPS 401).

- Add Rectified Aerial Photographs

    1. Go to the *14 - Grid Generation* folder, *Images*,

    2. Drag the image (.img) files over to the SMS window and drop where the plus sign appears by the pointer.

Scatter Set Conditioning

To line up and merge all scatter sets, some points may need to be deleted and or added. This is an essential yet tedious process in the development of a single file. Following the scatter set conditioning, all files will be merged into one file.

- Set Display Options to View All Points and Photo Background

1. Click on *Display*, *Display Options*, or the button,

2. Go to the *Scatter* tab in the Scatter options, change the points (Figure 4) to a small size and black,

3. Go to the *Contour* tab, and adjust the transparency to 50%.

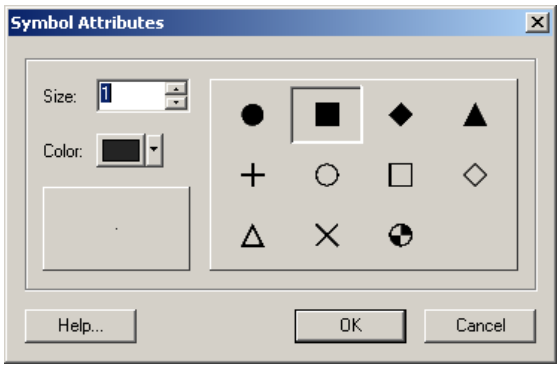


Figure 4. Points attributes.

Now the scatter sets can be viewed with the photo in the background.

- Identify Priority Datasets, Modify, and Merge

1. Use the button to select cells, delete button prompts the user whether or not they want to delete the selected points,

2. Select sections each file that will not be used in the final scatter set creation,

This process is largely up to the user's discretion and will require a trial and error method of generating a merged file to find inconsistencies in the merged dataset. There are several other methods to prioritize datasets during the merging process. For convenience, modified XYZ files are provided in the *XYZ-Modified* file.

3. Import new XYZ files, and delete original scatter sets by rt-clicking each old scatterset and selecting *Delete*.

4. Select *Scatter*, *Merge Sets*, select all sets, and make sure the *Merge All Scatter Points* is check on, click *OK* (Figure 5).

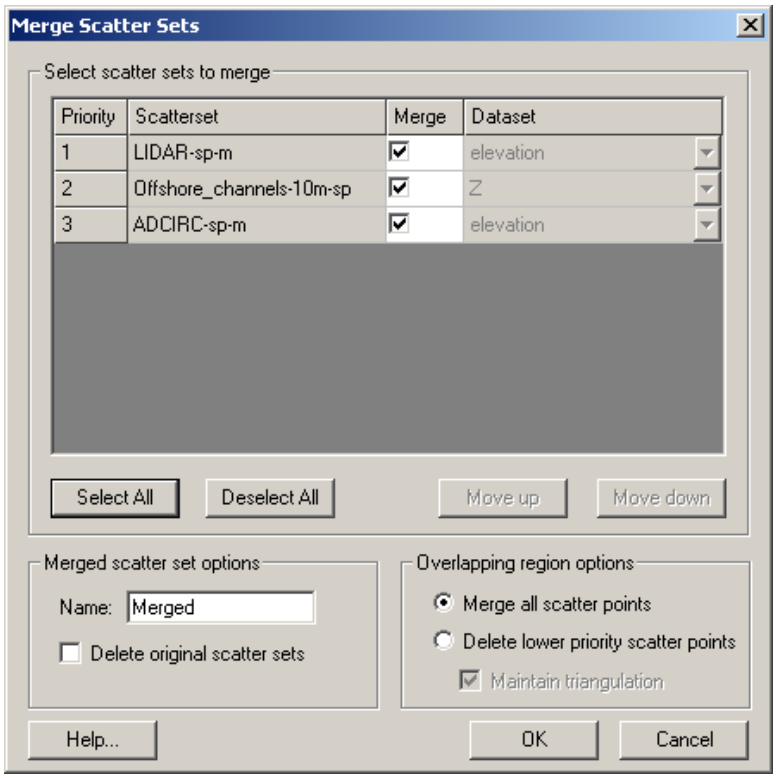


Figure 5. Merge scatter set options.

Be sure to save the project before and after making major changes. If the original files are deleted after they are merged, it is a good idea to have copies of all important changes made to the datasets being modified.

Generating the Grid

The CMS currently has two grid options it can calculate on: either constant, regularly spaced cells, or refined, rectangular, irregular cells. The refined option is valuable for optimizing (minimizing) CMS run time. Hydrodynamic and sediment transport time steps are dependent upon cell size in comparison to water depth and, therefore, an optimal number of computational cells are desired in combination with good resolution.

Though the main inlet channel is the primary area of focus for finer resolution, all channels carrying some proportion of current (>5% of the maximum velocities) should be represented by at least several cells in width. For inlet channel widths, a good rule of thumb is to have 10-20 ocean (computational) cells across smaller inlets (<300 m), and 20-50 cells across larger inlets (>300 m). For larger bays or river mouths with multiple entrance channels, the user should exercise discretion in maintaining the above suggested number of cells represented across each entrance channel. Further user discretion is advised for large slopes and bathymetry gradients to be represented by cell size.

The Humboldt Bay entrance channel is approximately 650 m wide, and a suggested number of cells across the channel is 30, resulting in approximately 20-25 m cell sizes.

- Set Refine Points for Grid Generation

    1. Click the *Map Module* button , or Map Data

    2. Rt-click on *default coverage*, and under *Type*, *Models*, select *CMS-Flow*,

    3. Click the *Feature Point* button to generate Refine Points, and place points in area of refinement,

    4. Click the *Feature Point Selection* button and select the generated point or points,

    5. Rt-click and click on *Attributes* (Figure 6), and set the base cell size for the area of the grid where the points are located.

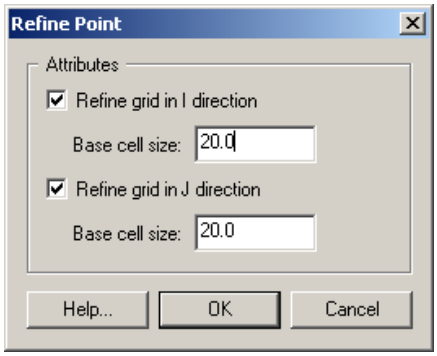


Figure 6. Refine Point options.

Next, the grid boundary is set square to general orientation of the coastline, with the computational direction, I & J (or IJ triad), set according to the model type. For the CMS-Flow coverage (which was selected in step 2), the point of origin will be automatically placed in the lower left corner of the grid after the grid has been created. This is default for CMS-Flow. The CMS-Wave IJ triad should be adjusted where the I-direction is pointing in the dominant wave direction, or from the open ocean. The model domain should be set to include all major reaches of the bay, extend laterally some kilometers from the inlet, and extend seaward well beyond the anticipated reach of the ebb jet. A good rule of thumb for setting the ocean boundary for inlet modeling is to place the boundary three or four times the expected length of the ebb jet.

- Generate the CMS-Flow Grid

    1. Click the *Create 2-D Grid Frame* button

    2. Draw two legs of the rectangular boundary for the model domain by clicking the lower ocean boundary corner point, then the lower landward boundary corner, and then the upper landward boundary corner to close the box.

3. Use the *Select 2-D Grid Frame* button to select the center point of the purple outlined grid domain, adjust the edges by dragging the center point of each edge, and change the orientation by selecting the at the corner of the grid,

4. Rt-click the highlighted center button and click *Properties* to further modify the dimensions of the grid boundary (Figure 7),

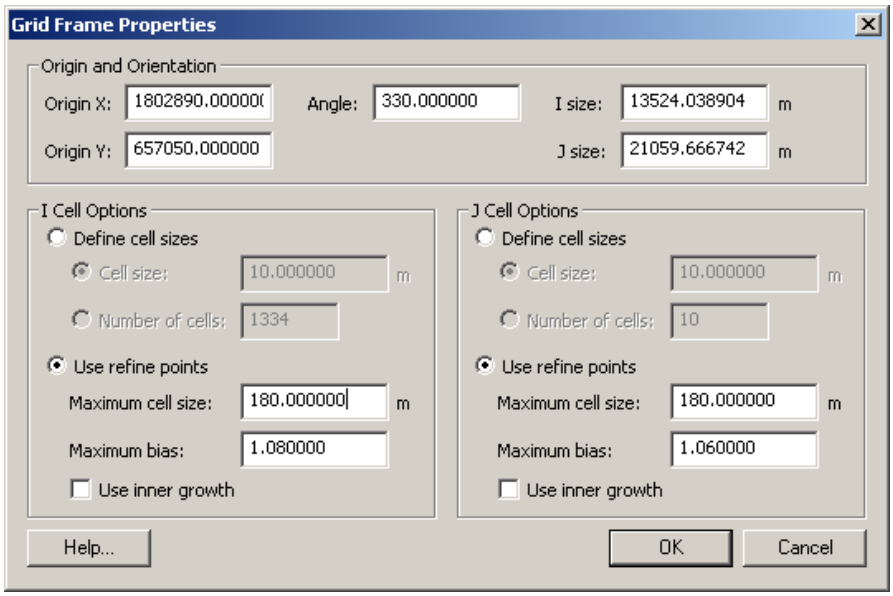


Figure 7. *Grid Frame Properties*.

5. Correct the angle orientation if it is far from 330º (the general orientation of the coastline), and set the I and J Cell Options to *Use Refine Points* and choose a maximum cell size and bias (cell size growth from refine points out), click *OK*,

6. Click *Feature Objects*, *Map→2D Grid*, and check that all the grid properties are set properly, *Use inner growth* is turned off, the *Interpolated Depth* (Figure 8) from the merged file is selected, and define an elevation cutoff where cells will be non-computational land cells (always dry, act as a wall).

Generating the first satisfactory grid typically takes some trial and error exploration to see how the grid looks. Be sure to modify the refine points to address any areas of concern, particularly the Humboldt Bay entrance channel, turning basin, and northern channel. Note that both the I and J directions do not need to be specified for each refine point, if refining only one direction is required.

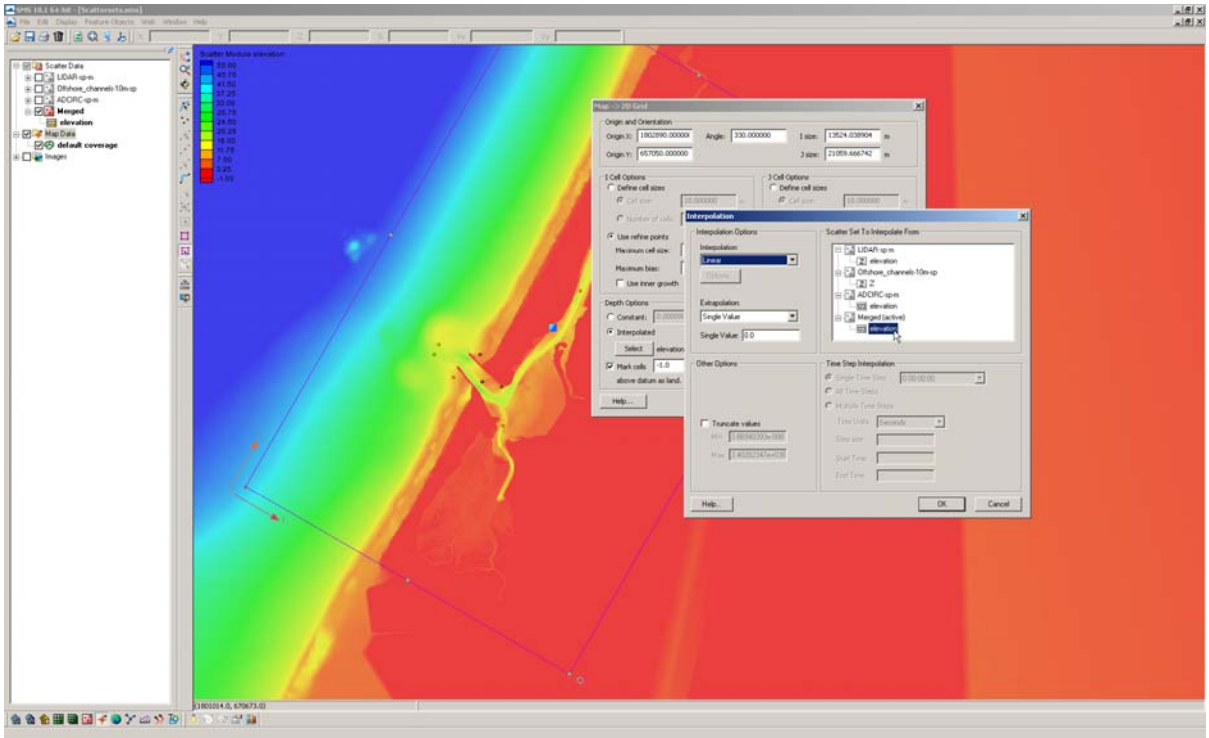


Figure 8. Depth Interpolation options for the Map→2D function.

After the grid is generated, two pop-ups warn of overlapping cellstrings and land-locked ocean cells. Cellstrings will be covered in section 14b, and so these can be left alone for now. The land-locked ocean cells that are not to be in the computation should be changed to inactive land cells. Also, the bay should be bound by land cells, and so any open ocean cells in the bay that lie on the grid boundary should similarly be changed to land cells. In addition, to allow for beach/berm cells to be wetted (cells with elevations higher than the land cell cutoff set in the grid generation section), sections of the beach should be changed to *Active Ocean* cells.

- Fix cells associated with warnings for the CMS-Flow Grid

    1. For changing land-locked ocean cells to land cells, click the *Select Grid Cell* button if not already highlighted, and change the Z number to -1.0 (Figure 9), or manually select cells by dragging the mouse over the target ocean cells and change the Z,

Figure 9. Toolbar at top of SMS window containing the X, Y, Z, and S (scalar quantity) of the selected cell or an average Z of multiple selected cells.

    2. Rt-click on the cells, select *Cell Attributes* (Figure 10), and change the cell type from *Active Ocean* to *Inactive Land* cells, *OK*,

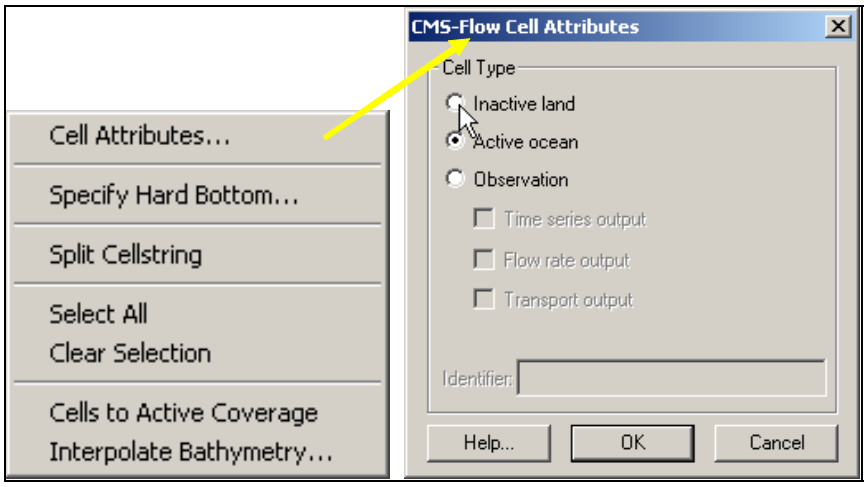


Figure 10. Toolbar at top of SMS window containing the X, Y, Z, and S (scalar quantity) of the selected cell or an average Z of multiple selected cells.

3. The same process can be done for ocean cells that need to be changed to land cells along the bay edges, and a simple way to check the entire land boundary is to use a directional selection method: click and hold down *ctrl* and drag the mouse along the outer edges of the landward side of the domain to highlight large sections,

4. To grab multiple areas, hold down shift while selecting new cells, and all will stay highlighted,

5. To change large sections of nearshore/beach cells that are land cells, select them with a polygon click clicking *Edit, Select with Poly* (clicking *Crtl + Shift* also initiates selecting by a polygon), and click a polygon covering a few cells across the ocean/land interface along a section of beach, and change to *Active Ocean* cells,

6. Change all required stretches of beach to ocean, and check that the landward grid boundaries have all inactive cells.

Save the project in a new folder to save the grid files (which are otherwise stored temporarily). Note that all files currently opened in the SMS are re-created in the new folder, and the older files are not modified.

Preconditioning Bathymetry

Due to the nature of bathymetric data collection resulting in random data that may be considered arbitrary (any given time) as compared to the time of your simulation period, certain areas of the grid may be poorly represented from scatter set triangulation and therefore the final generation of the grid. It is necessary to smooth some features of the grid and leave other areas intact. Presently, there is one option for smoothing grid bathymetry, and that is based on a weighted average of 3x3 or 5x5 cells in a rectangular selection of the grid. For Humboldt Bay, the nearshore and beach are the only areas that

need preconditioning of the bathymetry. The inlet, ebb-tidal delta, bay channels, and surrounding shoals with considerable relief should be left as close to the original bathymetry as possible (such areas are not arbitrary for coastal inlet simulations).

- Preconditioning the Nearshore

    1. Determine the bounding cell id by looking at the corner cells (that make up the rectangular area defining the nearshore that is to be smoothed) by clicking the *Select Grid Cell* button and clicking on each cell individually, the I and J id is listed along the lower portion of the SMS window (Figure 11),

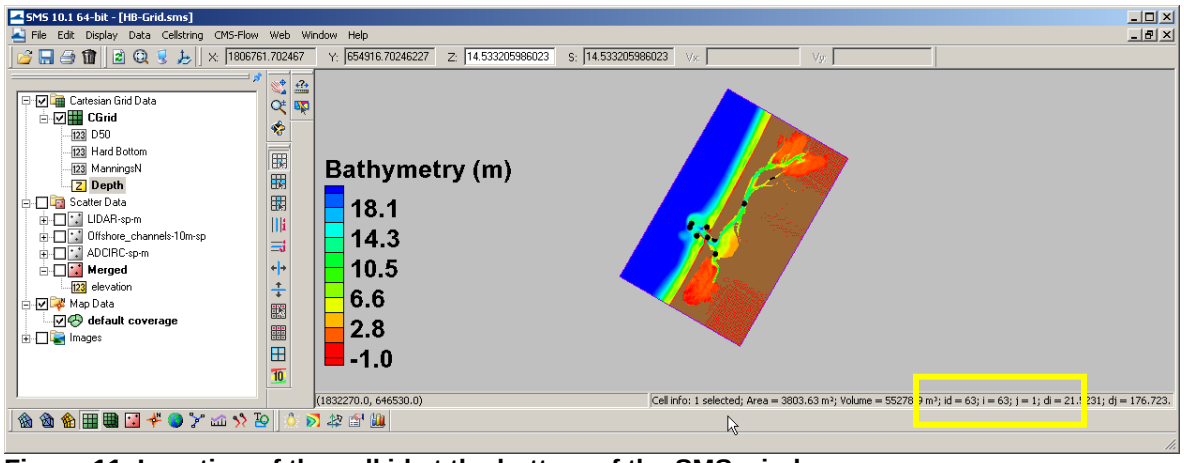


Figure 11. Location of the cell id at the bottom of the SMS window.

    2. The I direction is that of the columns and the J direction is that of the rows of the grid, record the I and J id of the bounding cells by step 1,

    3. Rt-click the *Cgrid* under *Cartesian Grid Data* and click on *Smooth* (Figure 12) and the *Cgrid Smoothing Options* will pop up,

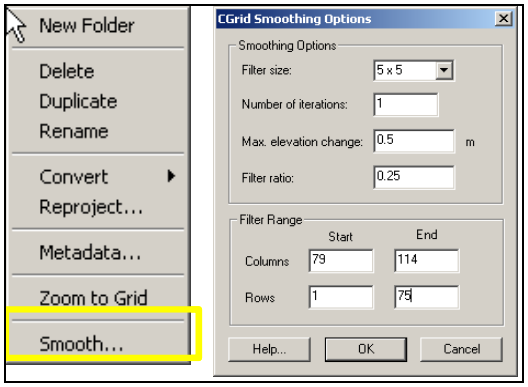


Figure 12. Drop menu of *Cgrid*; *Cgrid Smoothing Options*.

4. Set the Filter Size to 5x5, and leave the iterations, Max. elevation changes, and filter ratio to the default, and change the column's and row's *Start* ID and *End* IDs to what the bounding cells were (Figure 12), *OK*,

5. Check the area that was smoothed, repeat the process for the other stretch of nearshore bathymetry, preferably smoothing both the nearshore areas along the North Spit and the South Spit.

6. Save your grid with a new file name so you retain the original scatter set in case you need to repeat any pre-conditioning.

14.b.1. Hydrodynamics: Humboldt Bay, CA

Introduction

For this section of the hands-on tutorial, we will be using the base CMS-Flow grid as created in the Grid Generation section (14.a) previously covered. The present section will cover the following topics:

- Assigning Boundary Forcing information to cellstrings
  - Single water-level forcing curve
  - Water-level curve extracted from larger domain solution or tidal database
  - River-flow forcing curve
- Assigning Cell-specific parameters
  - Non-erodible depths (hard bottom)
  - Manning's N bottom friction
- Setting Model Control information prior to simulation runs
- Running CMS-Flow
- Loading solution and viewing results
  - Scalar contour and Vector options
  - Creating magnitude dataset
  - Generating animation of results

For this session, you will need the following files, located in the "Hands-on/Hydrodynamics" directory:

- HB-Grid_smooth.cmcards,
- HB-Grid-smooth_grid.h5,
- HB-Grid-smooth_mp.h5,
- Humboldt_WSE.xys
- Humboldt_flow.xys

Loading the Grid and Generating Forcing Cellstrings

Before the boundary conditions can be assigned, the grid must be loaded, and the appropriate cellstrings must be generated.

To load the grid, first select *File | Open* from the pull-down menu. Then browse to the "Hands-on/Hydrodynamics" directory and click the file "HB-Grid-smooth.cmcards" (shown in Figure 13).

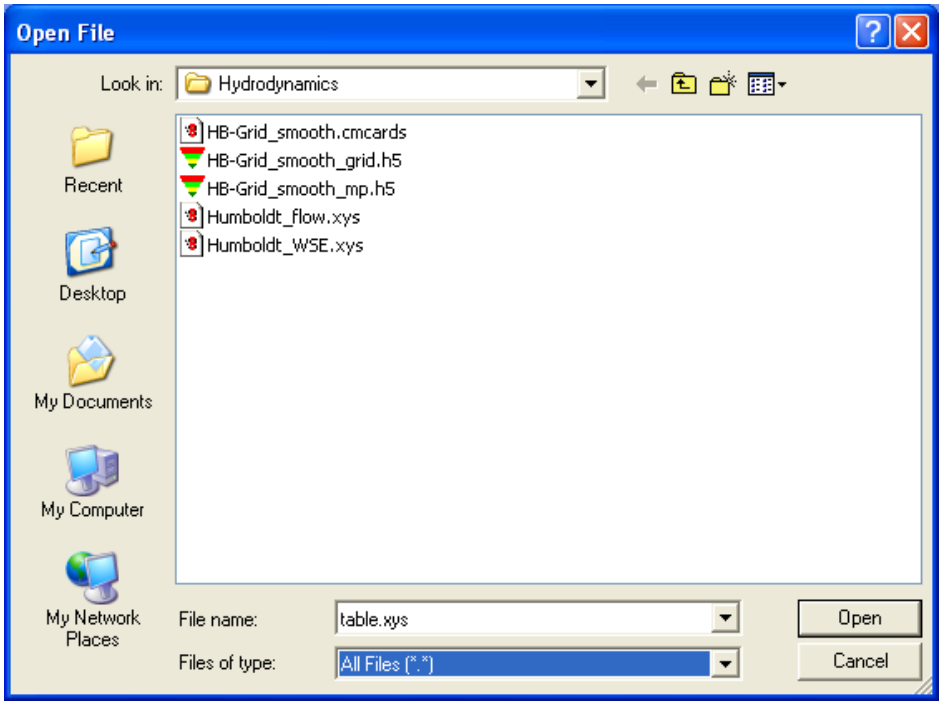


Figure 13. "File | Open" Dialog in SMS.

Within SMS, cellstrings are the vehicles by which boundary forcing information is stored for CMS-Flow grids and can be generated manually or automatically. Most of the time, it is easier to generate them automatically. To accomplish this, select *Cellstring | Generate Along Boundary* from the pull-down menu. This task generates not only forcing cellstrings, but cellstrings for land, as well. Later in this section, I will cover the manual creation of a cellstring for a river flow boundary.

For this session, we will not be using the land cellstrings and will delete them. Choose the "Select Cellstring" tool and select all but the ocean cellstring, as shown in Figure 14. Each cellstring has a selection box to assist in the selection process. Once all four land cellstrings have been chosen, press the "Delete" key, or click the Delete tool and answer "Yes" to the prompt.

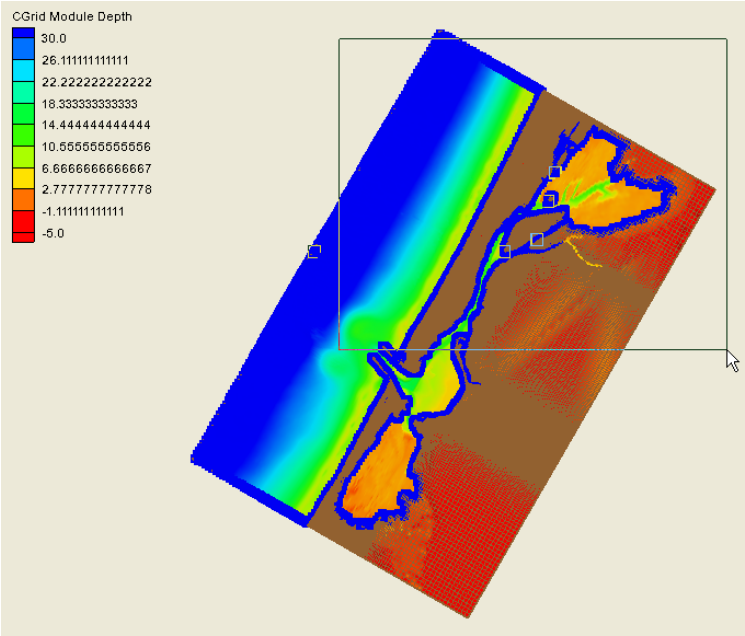


Figure 14. Selection of Land Cellstrings.

Assign Boundary Forcing Information

There are various boundary forcing types that can be assigned to each cellstring: flow rate (river flow), water surface elevation forcing (with or without velocities), and general constituent forcing (by amplitude and phase). Once each cellstring is selected, using the technique shown above for selecting cellstrings), the type of boundary forcing can be chosen. Select *CMS-Flow | Assign BC…*from the pull-down menu. The resulting menu is shown in Figure 15 below.

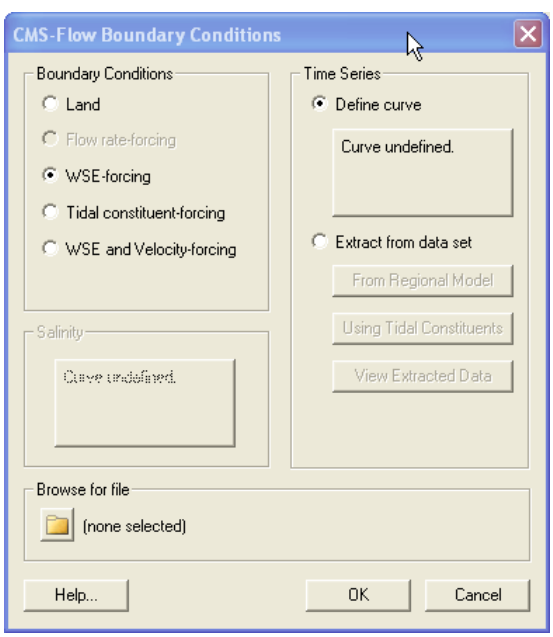


Figure 15. Boundary Condition dialog.

For this option, the choice made in Figure 15, should be "WSE-forcing," which opens additional choices on the right-hand side of the dialog. There are two water level forcing types, a curve that is constant for the entire cellstring, and a series of curves that varies from one end of the cellstring to the other. The variable forcing information is generated by extracting forcing information from some other source, such as:

- A larger regional model (such as CMS-Flow or ADCIRC)
- A tidal constituent database (such as the LeProvost global database or the ADCIRC regional databases)

To extract from a larger solution or from a tidal database, select the option "Extract from data set" and click either "From Regional Model" or "Using Tidal Constituents." If you choose the Tidal Constituent option, you must have downloaded and saved the appropriate type (LeProvost or ADCIRC tidal databases). Each database is available for download from http://www.aquaveo.com/downloads under the "SMS Downloads" section. If you decide to choose this option, make sure you have set the present projection (coordinate system), run start date, and run duration in model control, which will be demonstrated later in this section.

For this exercise, we will be assigning a constant water level from a time series file. To define a curve for this method, choose the "Define Curve" option from the Boundary Condition Dialog. Clicking the curve definition box will then bring up the time series editor shown in Figure 16 below, where a curve can be entered manually, cut and pasted from a spreadsheet, or imported from a file.

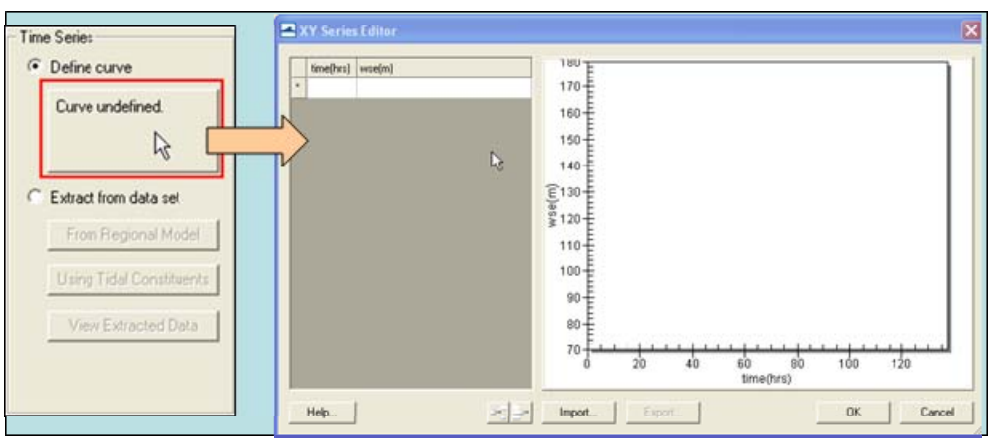


Figure 16. Time series selection.

Click the "Import" button and select the Humboldt_WSE.xys file which is located in the "Hands-on/Hydrodynamics" directory. Once the import is complete (as shown in Figure 17), click *OK* to close the time series editor and then click *OK* again, to close the Boundary Condition Dialog.

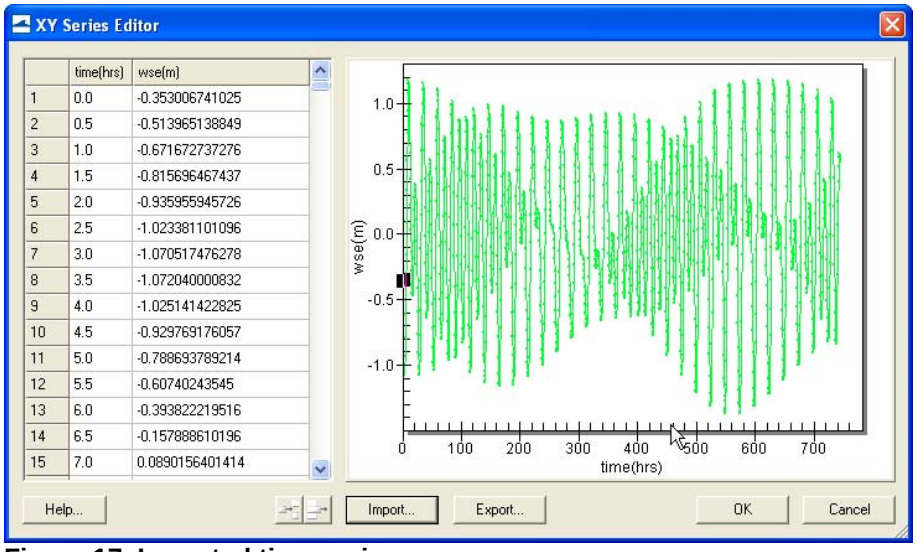


Figure 17. Imported time series.

Next, we will create a river flow boundary condition. We must first manually generate a cellstring for the river. Choose the "Create Cellstring" tool and click the first and last cells to be used for the River Flow boundary condition as shown in Figure 18. The order in which you select them does not matter. Next you must assign the forcing information for this cellstring. Enter the Boundary Condition dialog for this cellstring as you did previously. This time, choose "Flow rate-forcing" and open the time series editor. Import the flow rate forcing file named "Humboldt_flow.xys" from the same directory as before. The resulting curve is shown in Figure 19.

Notice that the values for this boundary condition are negative. This is because the forcing direction is relative to the X or Y direction from the point of origin, which is always the bottom left-hand corner of your grid. In this case, we have a river flow that is coming in from the East direction, so the flow rate values must be negative to ensure CMS-Flow forces that boundary in the appropriate direction.

As with most other major steps in building grids, you should SAVE your grid or project often.

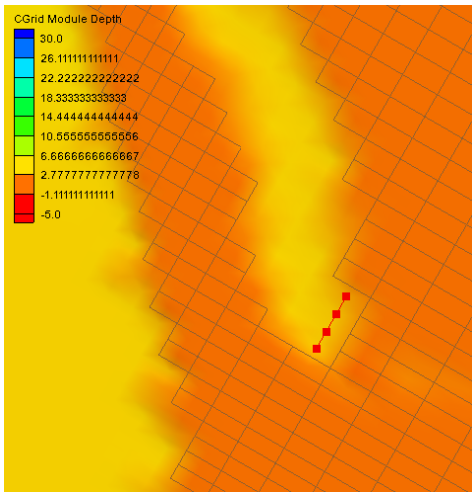


Figure 18. Create river flow cellstring.

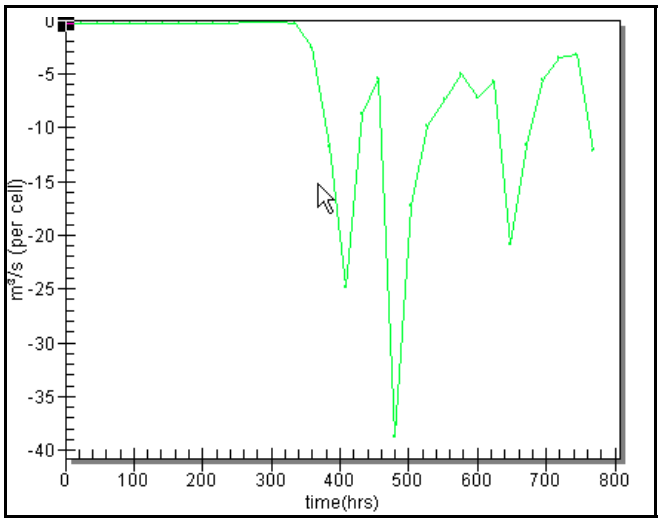


Figure 19. Flow rate curve.

Assigning Cell-specific parameters

Depending on the project area that you will be generating CMS-Flow grids for, there may be varying factors that you wish to represent in certain areas, such as rubble-mound jetties, a sea bottom which may be covered by non-erodible material (limestone or coral reef) or weirs. Additionally, at times a user may desire to represent locations where added friction is needed due to structures or increased turbulence due to sharp changes in current speed. There are methods available within SMS to represent these processes that are generally located in the interior of the domain, rather than the boundary of the domain.

Non-erodible bottom designation for selected cells

In the Humboldt Bay grid there are two jetties. A user may wish to designate both jetties and a surrounding area as non-erodible. To accomplish this, the user should select the cells with the "Select Cell" tool . To select multiple cells, hold the "shift" key down while clicking individual cells. You may also use a selection polygon, by holding the

"Ctrl" key down and clicking out a shape that encompasses the desired selection. Once the area is selected (similar to the area shown in Figure 20), right-clicking brings up a selection menu. Choose the option "Specify Hard Bottom" and the menu shown in Figure 21 will be displayed.

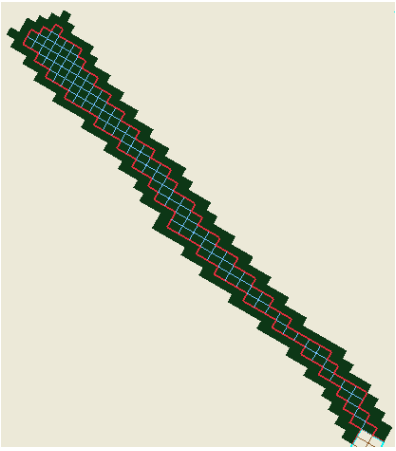


Figure 20. Selected cells for the Humboldt south jetty.

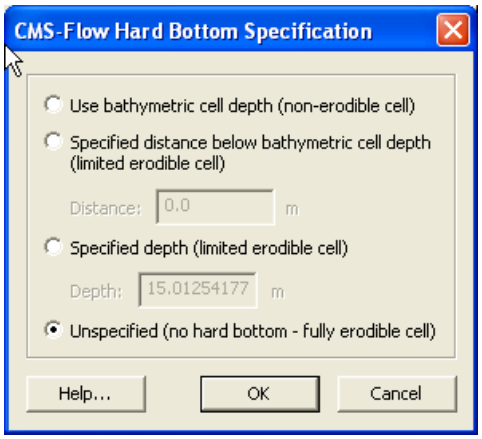


Figure 21. Right-click menu for selected cells.

There are several options on this menu:

- Use Bathymetric cell depth. This option assigns the Z value (depth) to be the maximum erodible depth. This is the most common choice.
- Specified distance below bathymetric cell depth. Use this option if you have an idea of the amount of sand above the hard bottom area.
- Specified depth. This option assigns the present assigned depth PLUS the entered value as the maximum erodible depth for each cell.
- Unspecified. This option designates all selected cells as freely erodible and is the default.

Once a choice has been made, click *OK*. If the user has selected to see hard bottom symbols, the screen will appear as shown in Figure 22.

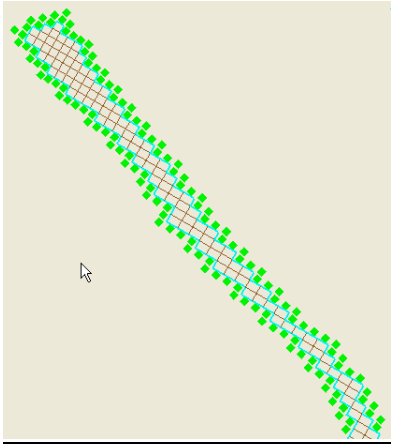


Figure 22. Hard bottom symbols for assigned cells.

If sediment transport is activated as a process for the CMS-Flow run, then these cells can erode down only to the assigned depth. Accretion of sediment is unaffected at these cells.

Bottom Friction (Manning's n) value assignment

A user may modify the bottom friction value assigned to certain cells if deemed necessary. The default value assigned upon grid creation is 0.025. The change from the default, a selection of cells must be made. The selection process is described in further detail in the Hard Bottom Designation section above.

1. Select the "ManningsN" dataset from the data tree. This dataset is a special "editable" dataset different than most others used by SMS.
2. Choose a selection of cells for which the bottom friction value is to be altered.
3. Change the value of the "S:" edit box (shown in Figure 23) and press "Enter." Note: only the first 9 significant digits are used. Disregard the remaining digits.
4. Each selected cells bottom friction value has now been modified and can be viewed with scalar contours if desired. Figure 24 shows a contour representation of bottom friction values for illustrative purposes only. No bottom friction modifications have been used for this basic Humboldt CMS grid.

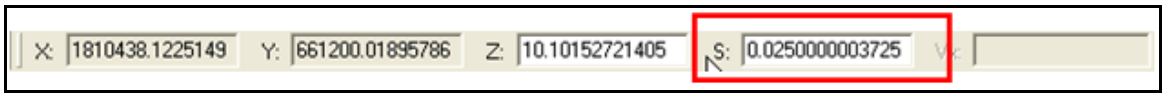


Figure 23. Scalar dataset edit box.

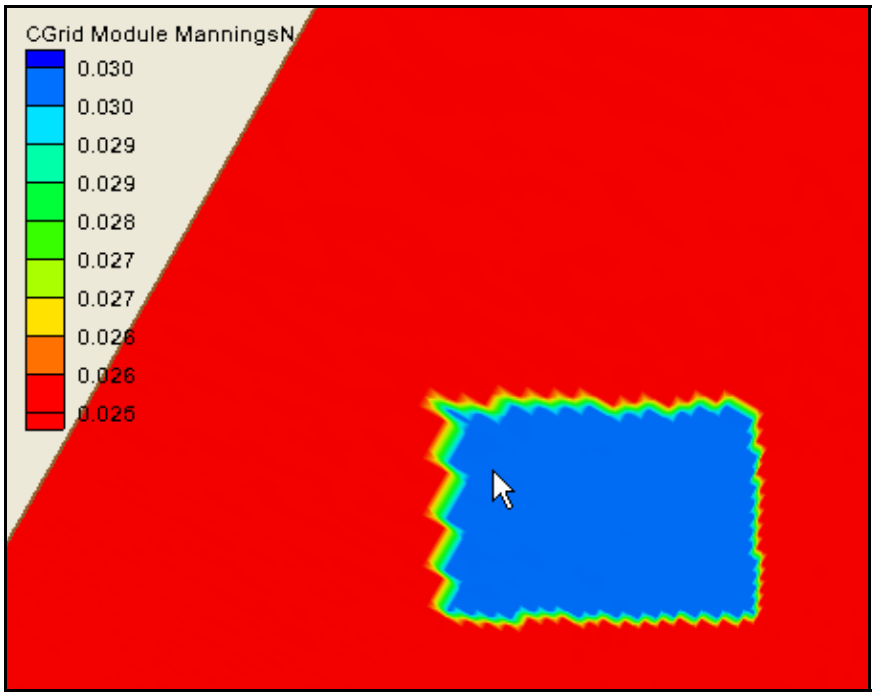


Figure 24. Contoured display of bottom friction values.

Setting Model Control

Various model parameters and options should be selected for any CMS-Flow simulation. These options are defined in the CMS-Flow model control dialog accessed from the *CMS-Flow | Model* Control pull-down menu. This section of the hands-on tutorial will go over the options which pertain to hydrodynamics. Other options will be discussed later.

The model control dialog (shown in Figure 25) consists of various pages (or tabs) of related information. This section will cover the following tabs:

- Model Parameters
- Wind/Wave
- Output
- Advanced

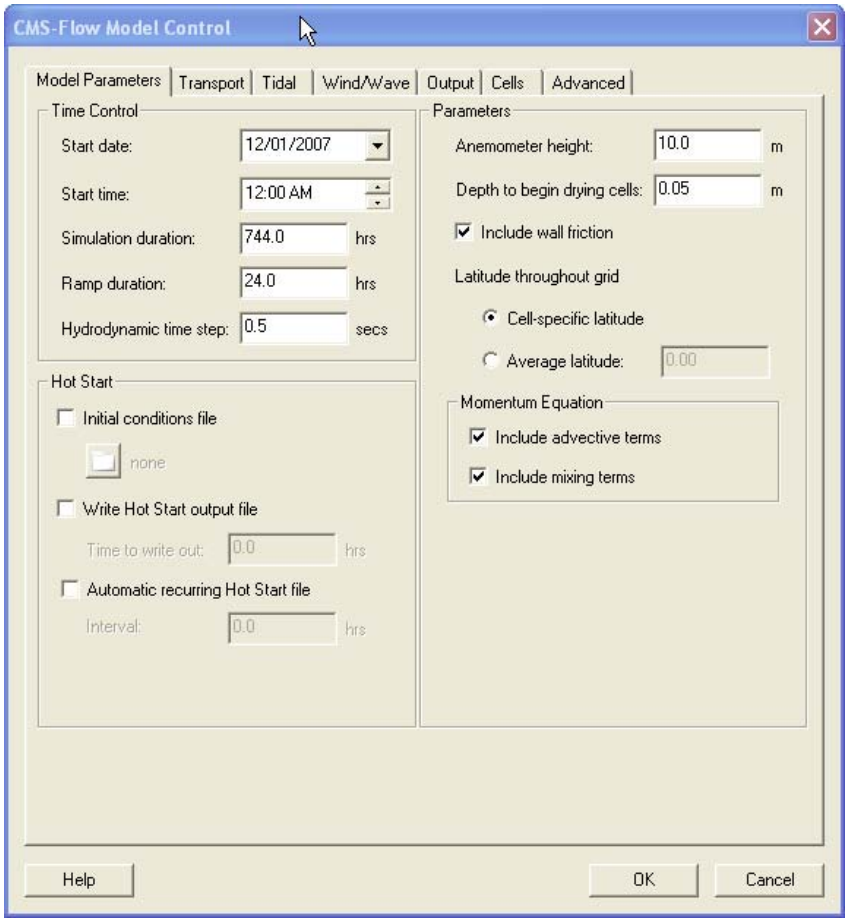


Figure 25. CMS-Flow Model Control.

Model Parameters Tab

- Start Date – Beginning date of simulation. Used in CMS-Flow to define the starting date of the simulation.
- Start Time – Generally, simulations should start at 0000 hr GMT, but could be changed to any time of the day.
- Simulation Duration – Number of hours that the simulation will run. In the future, an option to specify other units of time will be available (weeks, days, hours, minutes, etc.).
- Ramp Duration – The amount of time of reduced forcing to prevent instabilities brought on by the initial shock of the included forcing types (boundary forcing, wave forcing, wind forcing). All forcing types are reduced to zero at the beginning of the run and gradually increased to full forcing over the ramp period.
- Initial Conditions File – A "hot start" file can be specified to begin the simulation from previously-generated values. Generally this file would only be available if one of the next two options were used.
- Write Hot Start output file – CMS-Flow will write out a single hot start file after the entered number of hours have passed in the simulation.

- Automatic recurring Hot Start file – CMS-Flow will write out a hot start file at a specified interval (i.e. Every two hours)
- Anemometer height – Height that the anemometer was installed, if wind forcing has been selected.
- Depth to being drying cells – Minimum water depth at which a cell is designated as dry within CMS-Flow. Cells will re-wet once the conditions surrounding a dry cell are sufficient that the dry cell would have a depth of greater than this value.
- Include wall friction – Additional friction is included for cells next to land.
- Latitude throughout grid – Two options are available:
  - Cell-specific latitude – each cell has its own latitude value (default)
  - Average latitude – CMS-Flow assumes there is no variability in the latitude for a given grid.
- Include advective terms – Switch to turn on/off advective terms in the model run.
- Include mixing terms – Switch to turn on/off mixing for the model run.

Wind/Wave Tab

Wind data:

- Include wind – Switch to turn on/off wind forcing. Further options for wind are only available if this switch it checked.
- Import wind from file – Import wind information from one file that contains both velocity and direction information.
- Velocity (m/s) – Edit box to specify/import wind velocity information.
- Direction (deg.) – Edit box to specify/import wind direction information. Note: Directions are in "From direction" and follow the convention that wind directions start at zero from the north and increase in a clockwise direction (e/g/. Wind from due East is 90 deg).

Wave data:

- Include radiation stresses – Switch to turn on/off radiation stress forcing from a file mapped to CMS-Flow grid.
- Include wave conditions – Switch to turn on/off wave height, period, direction, and dissipation forcing from files mapped to CMS-Flow grid.

(NOTE: for CMS-Flow/CMS-Wave steering processes within the SMS interface, leave these unchecked. SMS will automatically update and provides the necessary files)

Output Tab

- Output Times Lists – Four separate available lists of output times. Each list can contain a separate list of output times that will be assigned to a certain type of dataset.
- Add by time step – Specify the Start and End time and an increment, then "Add" to the list which appears under the "Output Times" section. NOTE: Times added DO NOT have to be all at the same rate (e.g., Every 2 hours, etc).
- Output Datasets – Four types of datasets (Water surface elevation, Velocity, Morphology, and Transport), which can use any one, two, three or all four of the "Output Times Lists."

- Simulation Label – Label that will act as a prefix for the solution data when loaded. Can be used to separate various alternatives all generated for a basic grid geometry, so they can be loaded at the same time for comparison.

NOTE: Make sure you assign at least one output times lists to the available datasets in this section. If you fail to do so, your run will complete but you will have NO output to view.

Advanced Tab

The area on this tab is mainly for features developed for CMS-Flow that have not yet made it into the SMS interface. Rather than requiring multiple updates of the interface over a given period, this allows testing of new features without changes to the SMS graphical interface. A user can add a CARDNAME with options here, SMS will write the lines out to the parameter file, and CMS-Flow will run with the options (assuming it recognizes the CARD information.

One present application of this Advanced section is to enable use of multiple processes on your PC. The user should add a line for entry by using the "Insert new row above" tool , then add a line containing the following information to activate the feature within CMS-Flow, "OPENMP_THREADS ##" (do not add quotes). The ## should be replaced by a number from 2 to the number of threads on the computer which CMS-Flow will be run. The default without this card is to use only one thread.

Assigning Parameters for a Run

For this tutorial, set your model parameters how you want to, but you can use the following values as a guideline. I will only list parameters with non-default values.

- Model Parameters Tab
  - Start date: 12/01/2007
  - Simulation duration: 744 hrs
  - Hydrodynamic Time Step: 0.5 seconds
- Output Tab
  - Output Times 1
    - Start Time: 0.0 hrs
    - Increment: 0.5 hrs
    - End Time: 744 hrs
    - Click ADD
  - Output Datasets
    - Water surface elevation – Output Times 1
    - Velocity – Output Times 1
- Advanced Tab – Add one card "OPENMP_THREADS 2"

Make sure to save your grid, when you are through editing the Model Control dialog to write the parameters to the file.

Running CMS-Flow

At this point, a preliminary run of CMS-Flow can be made. To start CMS-Flow, select *CMS-Flow | Run CMS-Flow…* from the pull-down menu. SMS automatically runs a model checker specifically to check for potential CMS-Flow issues, which are shown in the "Problems" section as shown in Figure 26. For each issue, a brief description of the problem and brief fix is also shown. Many of these issues may not be severe enough to cause your model to fail; however, it is one indication of a possible problem.

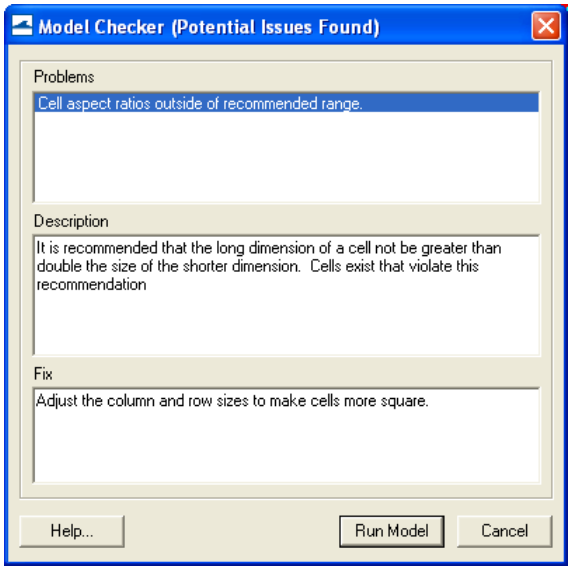


Figure 26. Model Checker dialog

Once your model has started, a CMS-Flow dialog box will appear with real-time messages from the model run (shown in Figure 27). Additional diagnostic messages can be found in the CMS-DIAG.TXT file that is created by CMS-Flow during the simulation run. An additional solution file (*_sol.h5) will be created as CMS-Flow is running. In the future, if the parameter file (*.cmcards) is loaded into SMS, the solution will be imported at the same time.

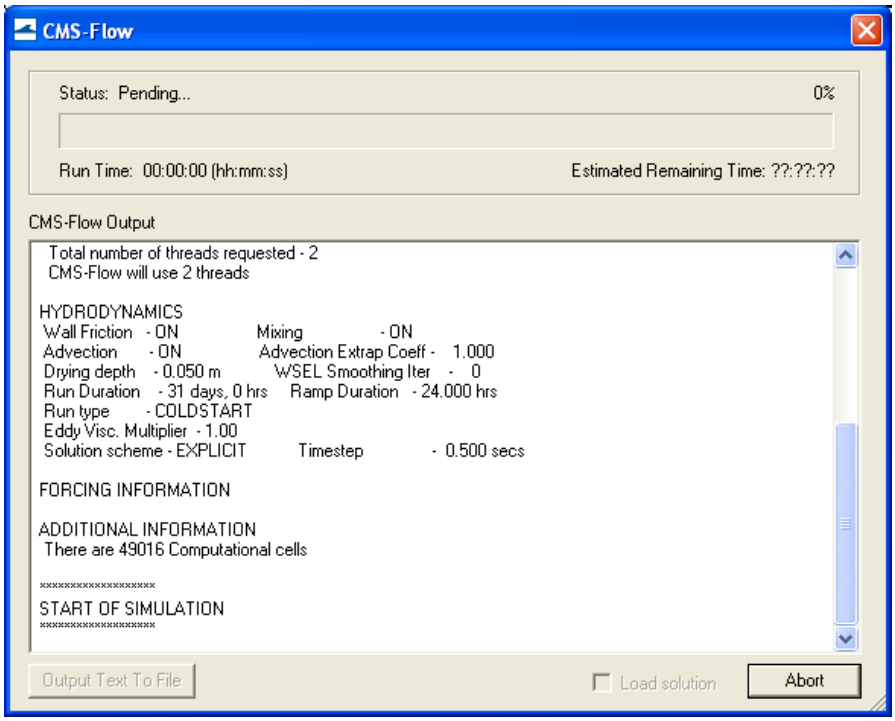


Figure 27. CMS-Flow run screen

Loading solution and viewing results

For the purposes of this hands-on tutorial, we will be loading a pre-calculated solution. Before loading the completed solution, we want to delete all of the presently loaded data in SMS. To clear the data and load the completed solution:

1. Select *File | Delete All* and press *Yes* to confirm.
2. Select *File | Open* and browse to the "Hands-on/Hydrodynamics/Solution"
3. Click the file named "HB_Workshop.cmcards" and press *Open*.
4. Select *Display | Display Options*, and make sure the "Contours" option is checked.

This grid will look slightly different than the grid used in the previous section because it is based on a different bathymetry set. The method for viewing the results is the same, however. Notice that the solution for this grid set up is loaded in addition to the grid (see Figure 28).

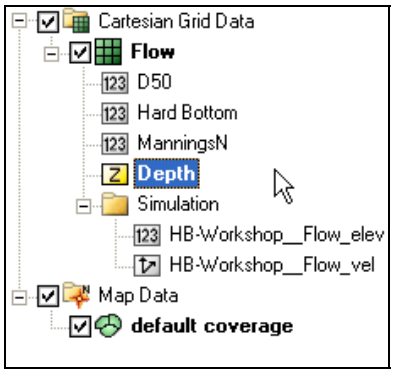


Figure 28. Simulation dataset in data tree

Viewing Contours for individual datasets

Each scalar dataset can have its own scalar contour options defined. To demonstrate, we will be modifying the water surface elevation dataset. Right click the dataset named "HB-Workshop__Flow_elev" in the data tree and choose "Dataset Contour Options." This will bring up the Dataset Contour Options dialog (shown in Figure 29) where a user can modify various aspects of how that dataset is to be displayed in SMS.

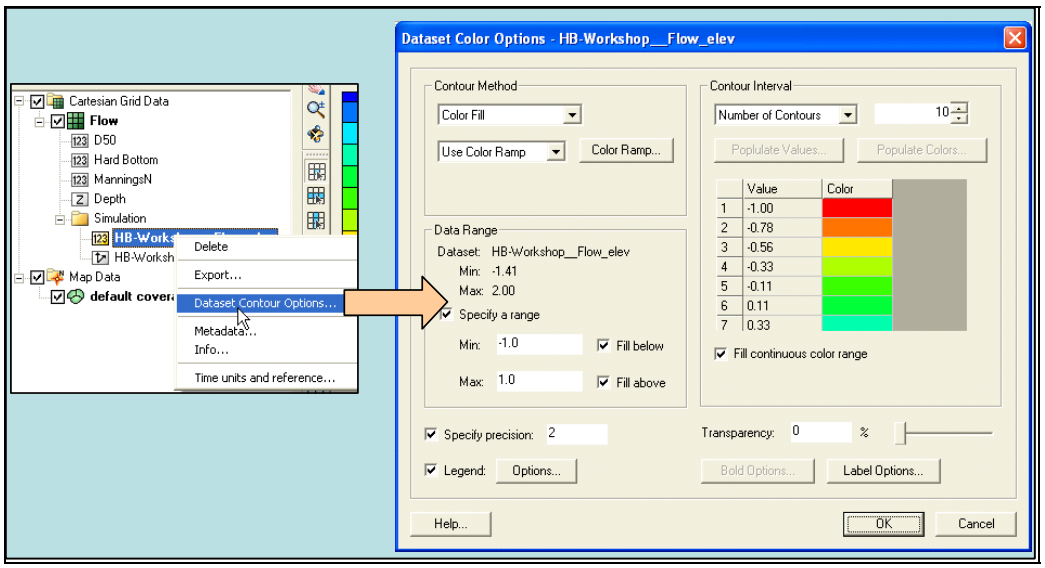


Figure 29. Dataset Contour Options

The sections of this dialog that can be altered are listed below.

- <u>Contour Method</u>
  - Fill type – Three choices: "Linear only," "Color Fill." or "Color Fill and Linear"
  - Color options – Two choices: "Use Color Ramp," or "Specify Colors"
    - "Use Color Ramp" has an additional dialog to alter the color palette.
- <u>Data Range</u>
  - Dataset – Lists the active dataset name and the Min/Max range
  - Specify a Range – Manually enter the minimum and maximum values for the range.

- Contour Interval – Three choices: “Number of Contours,” “Specified Interval,” or “Specified Values”
- Specify Precision – Enter the number of digits of precision to be displayed in the contour legend
- Legend – Turn on/off the Contour legend, and specify “Options” for that legend.
- Transparency – Enter a percentage of transparency for the contour display. Useful to allow the user to see through to a loaded image or another dataset.

For this tutorial, we will use the following settings:

1. Contour method: Color Fill, using a Color Ramp with the following options
    a. Hue Ramp
    b. Do not reverse the color scheme.
2. Data Range: Check the “Specify a Range” box and enter the values -1.00 and 1.00 as the minimum and maximum value respectively.
3. Contour Interval: Number of Contours – 10
4. Specify precision: 2 digits of precision
5. Legend options:
    a. Title – “Water Surface Elevation, m”
    b. Equal color segment height box – checked
    c. Height – 200 pixels and Width – 20 pixels.
6. Transparency – 0 percent.

When finished editing options, press *OK*. The resulting screen should appear similar to what is shown Figure 30. Once the contour options are set to the desired look, the user may examine individual timesteps which were saved by the numerical model. The user can scroll through the times listed in the Time Steps window (shown in Figure 31)

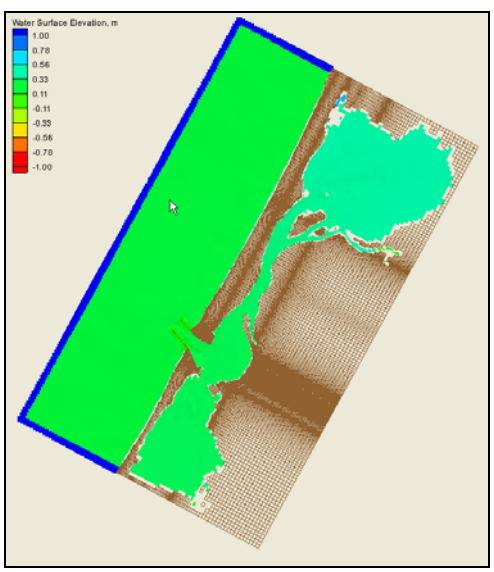


Figure 30. Water Surface Elevation dataset contours

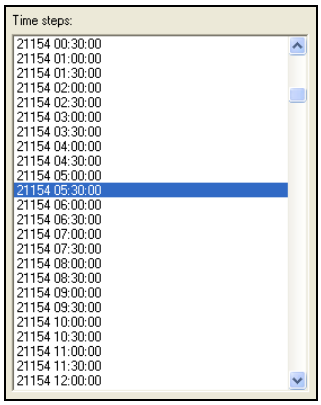


Figure 31. Time steps window display

Viewing Velocity Vectors

When viewing a vector-based dataset, such as the one saved for current velocity, you will first need to turn on the display of those vectors. Select *Display | Display Options*, and make sure the "Vectors" option is checked. In the data tree, select the velocity dataset named "HB-Workshop__Flow_vel." The initial display of the velocity dataset vectors, shown in Figure 32, demonstrates the need to change the appearance of the vectors. There are too many vectors shown on the screen to make sense of what is going on. A better option would be to show normalized vectors only in areas of significant current speed.

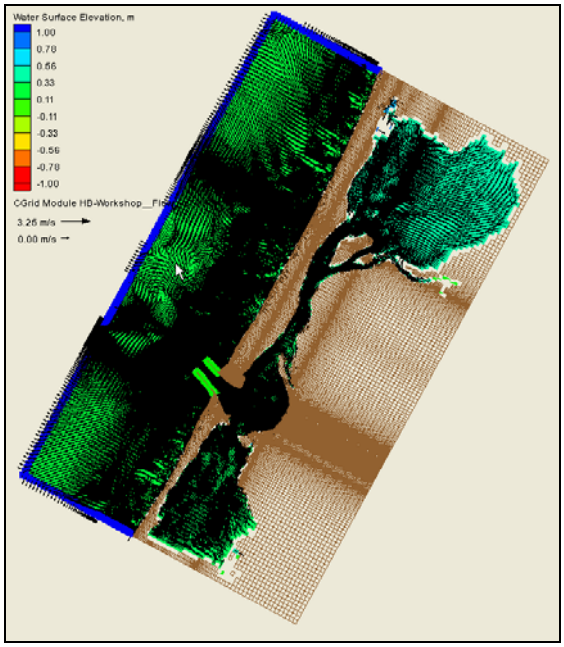


Figure 32. Velocity dataset display

To change the velocity vector appearance, the user can either select *Display | Display Options* and click the "Vectors" tab, or click the "Vector options" tool which is normally located along the bottom of the screen on the Macro toolbar. This action will

bring up the Vector Options dialog shown in Figure 33. The options typically modified are:

- <u>Vector Display Placement and Filter</u>
    - Display – Allows the user to change the placement of shown vectors by the four options: “at each node,” “at corner nodes only,” “on a grid,” and “on a coverage.” Typically the user will select either “at each node” or “on a grid.”
    - Z-offset – Raises the vector display layer the specified distance above the grid. This is useful, when the scalar dataset presently selected hides some of the vectors.
    - Data Range – Two choices: “All” or “Range.” At times there may be a display issue which shows extremely long vector arrows. This issue can be corrected by changing this option to “Range” and entering values for the Minimum and Maximum.
- <u>Legend</u>
    - A check box to turn the display of the vector legend on/off.
    - An options dialog to alter the way the legend is displayed.
- <u>Arrow Options</u>
    - Shaft Length – Three choices: “Fixed length,” “Scale length to magnitude,” and “Define min and max length.” These choices change the appearance of the length of the vector arrows.
    - Color Range – Options are available for the coloration of the vectors.

When the user has completed the modifications to this dialog, they should press *OK*.

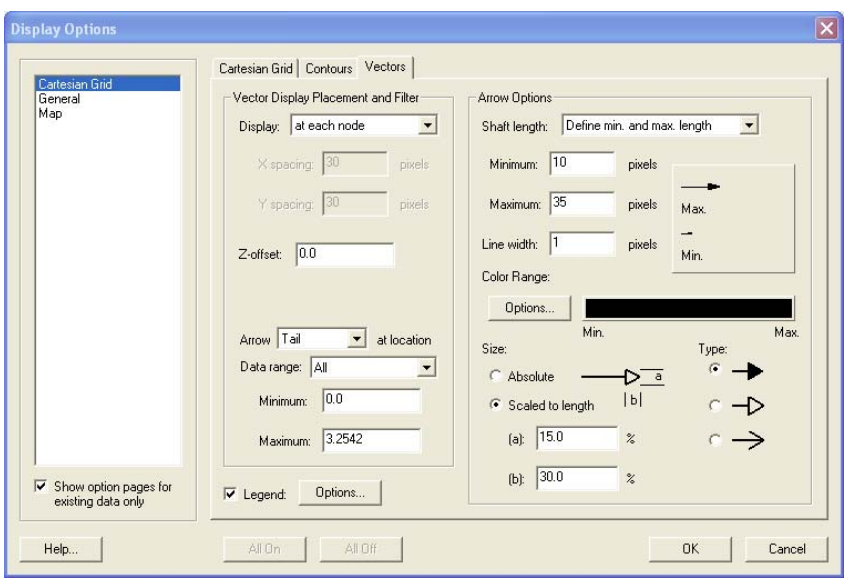


Figure 33. Vector options dialog

For this tutorial, we will use the following settings:

1. Display: “On a grid,” using X/Y spacing of 20 pixels each.
2. Z-offset: 1.0

3. Data range: “Range”
    a. Minimum – 0.0
    b. Maximum – 2.0
4. Legend: Unchecked
5. Shaft Length: “Scale length to magnitude” with ratio of 25.0
6. Remaining options left at default value.

The resulting display should look very similar to what is shown in Figure 34.

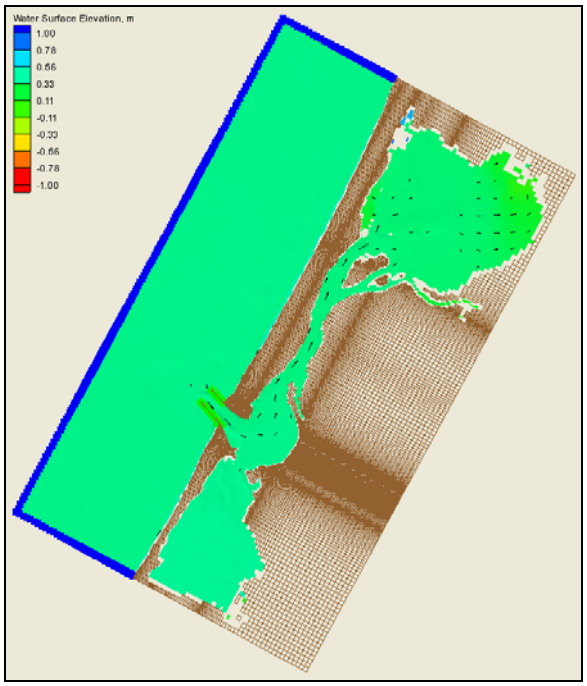


Figure 34. Vector display after editing

Create a velocity magnitude dataset

At times, a user may want to view scalar contours that relate to the magnitude of the current speed. A scalar dataset for magnitude can be created from the velocity vector dataset with the following steps:

1. Select *Data | Dataset Toolbox* from the pull-down menu. The resulting dialog is shown in Figure 35.
2. Click “Vector to Scalar” under the “Conversion” section of the “Tools” window.
3. Click the velocity vector dataset shown under the “Data sets” window.
4. Check the “Magnitude” check box under “Options.”
5. Give a name for the resulting dataset, such as “Velocity.” Note: The option name “Magnitude” will be added to the end of this prefix when this new dataset is added to the data tree.

6. Press *Compute* and wait for the process to complete. A status is red is shown on the bottom-most line of the SMS main window.
7. Press *Done* and verify that the new dataset has been added to the data tree.

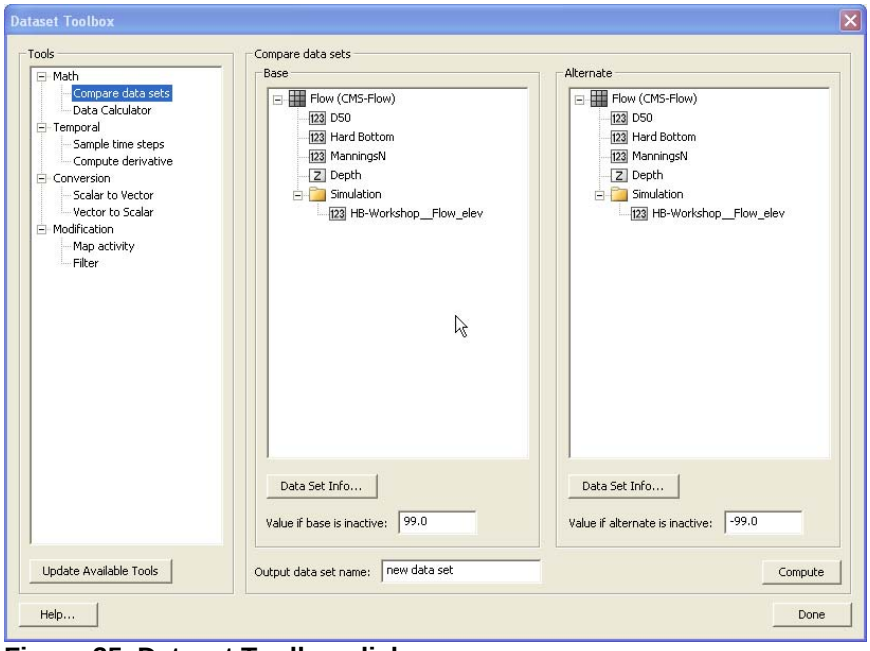


Figure 35. Dataset Toolbox dialog

Use the techniques defined in the "Viewing contour" section above to change the appearance of the "Magnitude" dataset contours. We will use the following settings:

1. Contour Method: Color Fill using Color Ramp with the following options
    a. Select Hue Ramp
    b. Click "Reverse" button to reverse the colors. Typically when showing velocity magnitude, slower speeds are designated with cooler colors and faster speeds are designated with warmer colors.
2. Specify a range: **Checked** and enter values of 0.0 for minimum and 2.0 for maximum.
3. Specify precision: **Checked** and enter value of 2
4. Legend: **Checked** and for the options:
    a. Title: Current speed, m/sec
    b. Height 200 pixels and Width 20 pixels
5. Leave the remaining options with their default values.

The resulting screen will appear similar to what is shown in Figure 36.

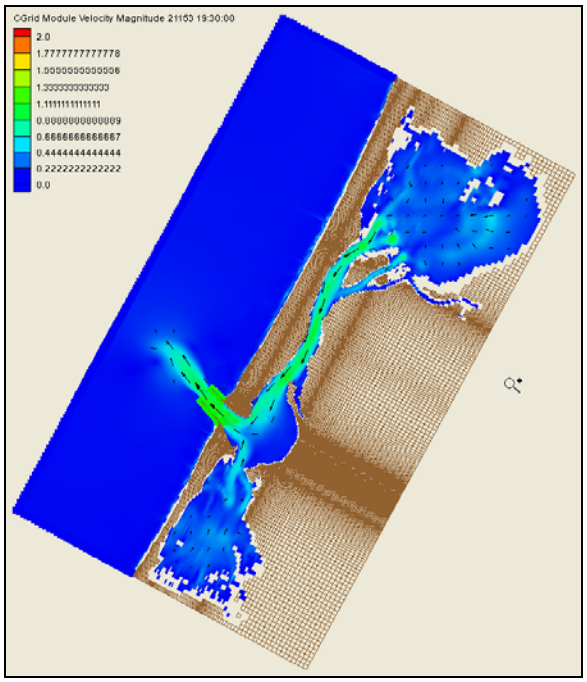


Figure 36. Velocity Magnitude display

Generating an animation

After setting up the display to the user's specifications, an animation can be made of the solution through time. The steps to accomplish this in the SMS are listed below:

1. Ensure that the desired scalar and vector datasets are selected.
2. Select *Data | Film Loop* from the pull-down menu. The resulting dialog is shown in Figure 37.
3. Check the box beside "Create AVI file" and click the button, to assign an animation file name.
4. Click the *Next* button, to proceed to the next screen shown in Figure 38.
5. Modify the length of time to animate. For this example enter the number 7 and change the units from "hours" to "days."
6. Select the option "Specify Time Step Size" and enter the value 0.5 and leave the units as "hours."
7. Click the *Next* button to proceed to the next screen shown in Figure 39.
8. Click the *Clock Options* button to enable a clock.
    a. Select "Bottom Left Corner"
    b. Check the box "Show progress bar"
    c. Change the Type to "Horizontal" and enter a Size of 40.
    d. Check the box "Show the time"
    e. Press the *OK* button to return.
9. Press the *Finish* button to generate the animation and save the file.

The animation automatically starts playing after it has finished generating.

This concludes the hydrodynamics section of the tutorial.

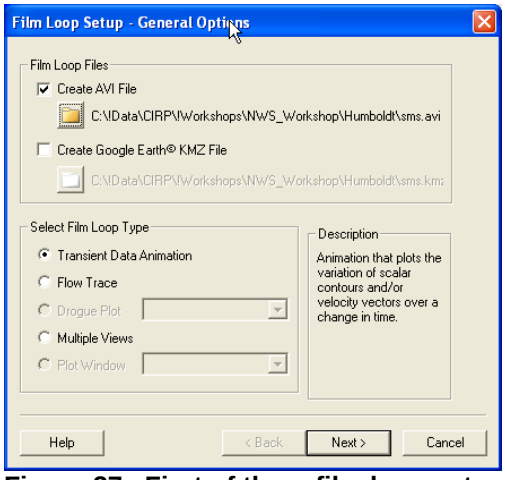


Figure 37. First of three film loop setup dialog screens

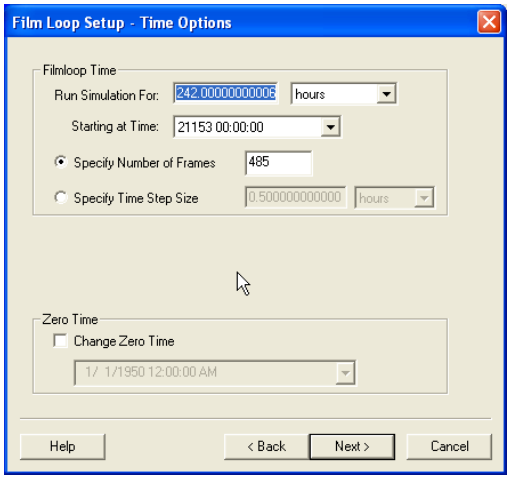


Figure 38. Second of three film loop setup dialog screens

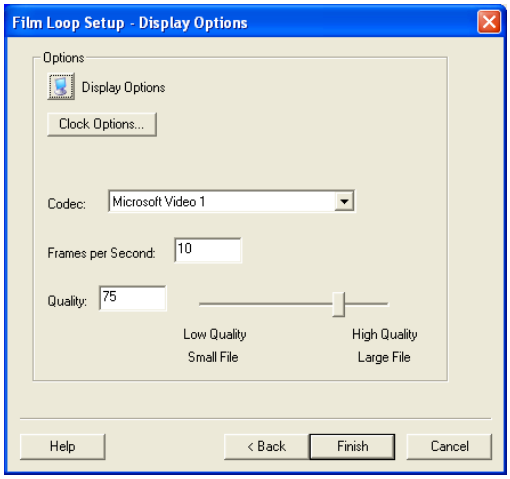


Figure 39. Third of three film loop setup dialog screens

14.b.2. Sediment Transport / Morphology Change: Humboldt Bay, CA

Introduction

For this section of the hands-on tutorial, we will be using the base CMS-Flow grid as created in the Grid Generation section (14.a) previously covered. The present section will cover the following topics:

- Assigning Variable D50 values
- Setting Model Control information prior to simulation runs
    - Choosing sediment transport algorithm
    - Choosing sediment transport parameters for NET
- Loading solution and viewing results
    - Scalar contour options for morphology
    - Creating morphology change dataset

For this session, you will need the following files, located in the "Hands-on/Morphology" directory:

- HB-Workshop__Flow.cmcards,
- HB-Workshop__Flow _grid.h5,
- HB-Workshop__Flow _mp.h5

Variable D50

When sediment transport and morphology change are activated, sometimes it may be desired to designate areas of the grid to have a certain grain size. The present version of CMS-Flow is not able to track individual grain sizes with the flow; however, as a first phase approach to this problem, CMS-Flow is able to designate areas of the grid to have the characteristics of a certain sediment grain size. The default sediment size that SMS assigns each grid is 0.2 mm.

To try this out, we have set up our Humboldt Bay grid, to have five (5) zones of sediment sizes which vary between 0.2 to 0.5 mm. This values used in this grid set up are only an example of how to use this feature and is not indicative of any research done for this region.

The ability to change grain sizes is very similar to section on varying the bottom friction (Manning's N) value in the previous tutorial. After selecting the D50 dataset in the data tree, the user would select a series of cells by any method and alter the D50 value for that selection. See Figure 40 for the set up that has been generated by CHL.

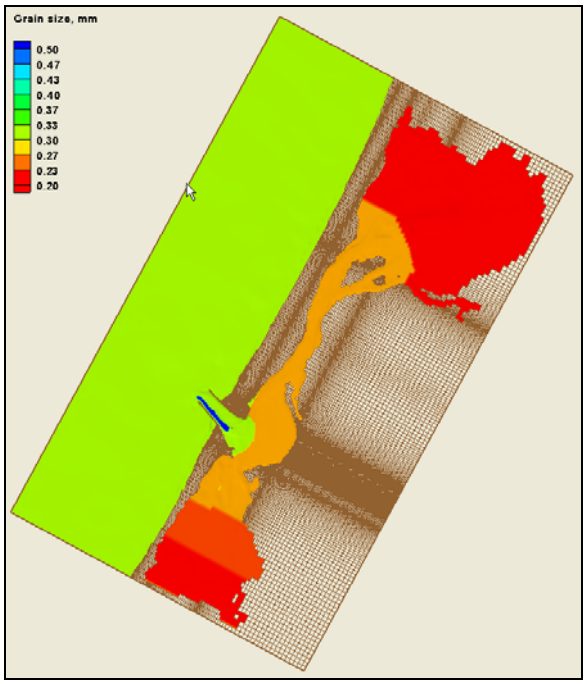


Figure 40. Variable D50 values

Setting the Sediment Transport Parameters

Sediment transport must be turned on in the Model Parameters in order for morphology change to be calculated. At the same time, the results of sediment transport and morphology calculations will be lost unless the output is designated for each. For the present version of CMS-Flow, which is an explicit model, timestep values are small (on the order of seconds) and therefore setting the output intervals to as small a scale as minutes will work. However, the time step for morphology (set in the *Model Parameters* tab) will limit the necessary interval for morphology output.

All output datasets are optional, and so some user discretion is advised when setting the output intervals to small increments over long periods of time as this will result in very large files (10s of gigabytes). For this example, though, 744 hours will not generate more than a 2 gigabyte solution file with the below output setup.

- Setting the Output Time Series for Sediment Transport and Morphology

    1. Go to *CMS-Flow*, *Model Control*, and click on the *output* tab,

    2. Set the time series output for the *water surface elevation* and *velocity* to the same output interval of 30 minutes, similar to Figure 41,by highlighting *Output Times 1*, setting the *Start Time* to 0.0 hr, *Increment* to 0.5 hr, and *End Time* to 744.0 hr, and clicking *Add*,

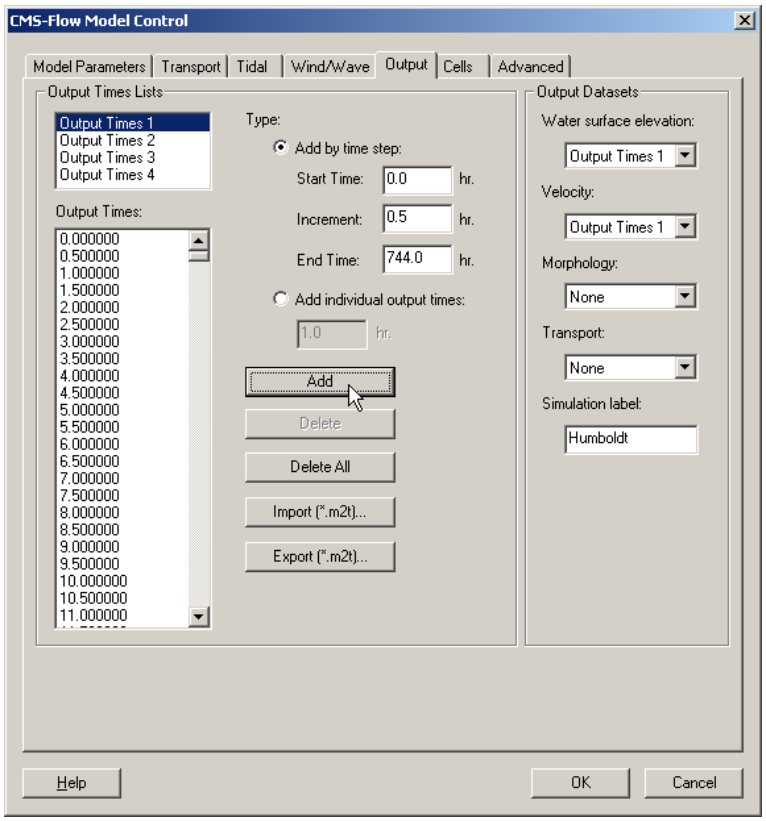


Figure 41. CMS-Flow Model control: Output tab for specifying time series output of CMS-Flow calculated datasets. Following Step 2.

3. Set *Output Times 2* for *Morphology* output at a 3 hour interval, by switching to , and changing the *Increment* to 3.0, and selecting that on the drop down menu for Morphology

4. The same can be done for *Transport*, which should normally be set to an *Increment* on the order of the tide or wave time series (<3hrs; typically 0.5-1.5hrs), and it is suggested that it be set to 1 hour output increment for *Output Times 3* (Figure 42), *OK*.

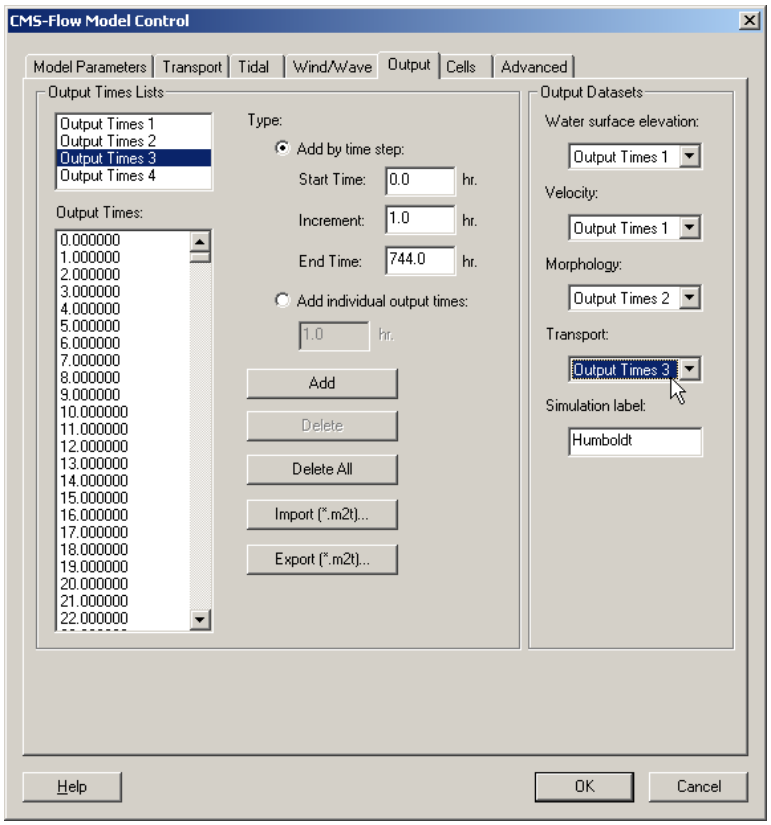


Figure 42. CMS-Flow Model Control: Output tab for specifying time series output of CMS-Flow calculated datasets. Following Step 4.

Sediment Transport options will be set to the default (Figure 43) when the *Calculate Sediment Transport* box is checked, and the current default will automatically check *Use non-equilibrium transport*. This represents the default settings and do not represent the suggested settings for calculating sediment transport. It is important that these settings be specified depending on the location and be calibrated to field measurements. Unchecking the Non-equilibrium transport (NET) option will turn NET off and allow for the selection of either the Total Load or A-D *Formulation.*

The user has the option to choose for the *Transport Formula* (*Transport Capacity Formula*): Lund-CIRP, Van-Rijn, or Watanabe. The default *Sediment density* is based quartz sand, and default *Water density* is based on a typical ocean water. *Sediment Porosity* of rounded quartz sand is approximately 0.4.

The *Bed load* and S*uspended load scaling factors* directly multiply by the values given by the transport formulas mentioned above. The Bed Slope coefficient accounts for the change in direction of the sediment transport due to the bed slope. The default value is 1.0 but should be calibrated to field measurements.
In order to properly simulate the sediment transport, calibration with field measurements is always necessary. The field measurements may be suspended concentration measurements, local transport rate, or morphologic change. See the Sanchez and Wu (2009) for details of the NET specifications.

- Setting up the Sediment Transport and Morphology Change

  1. Go to *CMS-Flow*, *Model Control*, and click on the *Transport* tab,

  2. Turn on sediment transport by checking the *Calculate sediment transport* box, and make sure the *Use non-equilibrium transport* is also checked, and set the *transport time step* to 0.4 seconds and the *morphology time step* to 20 seconds,

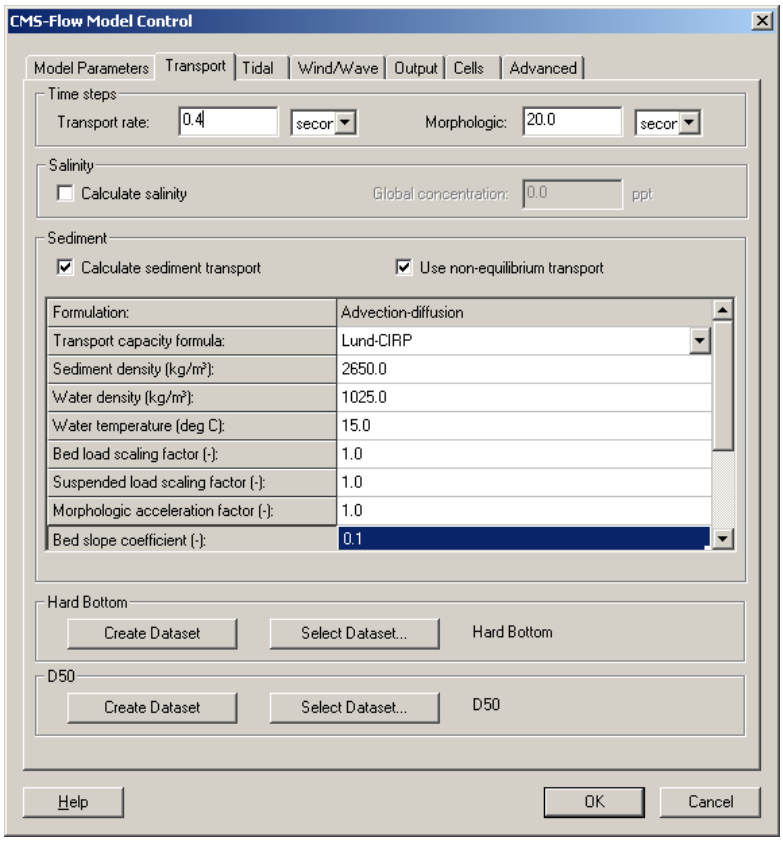


Figure 43. CMS-Flow Model Control: Transport tab for specifying transport and morphology timesteps, Global Salinity calculation, and Sediment Transport Parameters.

  3. Set the *Sediment* parameters to those indicated in Figure 44, of which all but Bed Slope (0.1) are set to default parameters,

  4. Figure 44 illustrates the bottom parameters to set for the NET & Advection-Diffusion formulation setup, which include the adaptation formulation and length,

5. The Hard Bottom and D50 should be set to the datasets defined in the grid datasets, and can be checked by clicking Select Dataset, and selecting the appropriate scalar dataset of the grid.

At this point you can save your grid and run CMS-Flow.

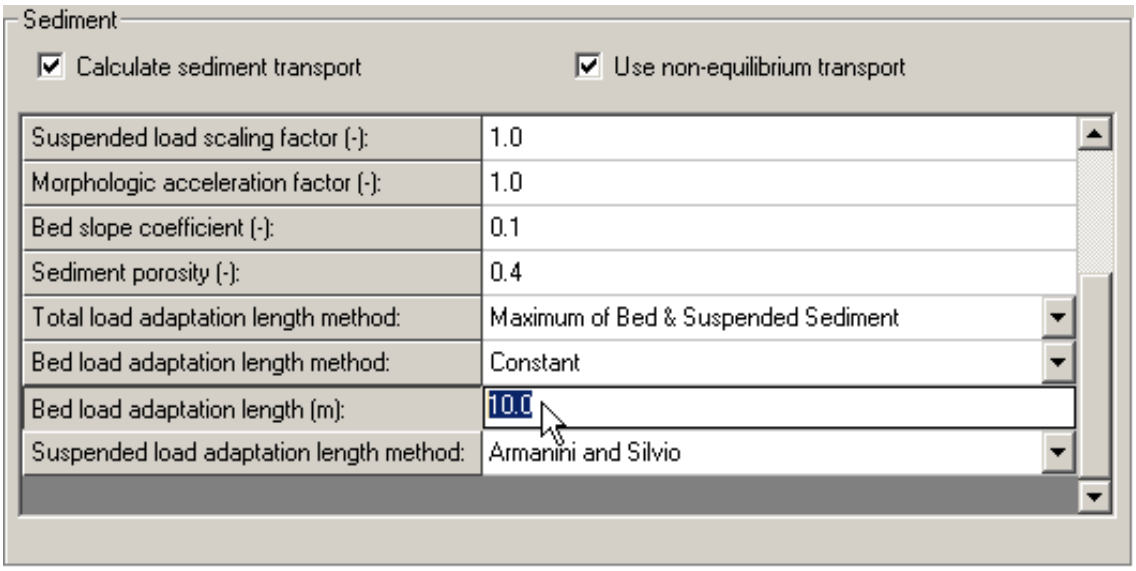


Figure 44. Lower section of Sediment transport parameters. Note the Bed load adaptation length method can be a constant length or depth dependent.

Loading solution and viewing results

For the purposes of this hands-on tutorial, we will be loading a pre-calculated solution. Before loading the completed solution, we want to delete all of the presently loaded data in SMS. To clear the data and load the completed solution:

5. Select *File | Delete All* and press *Yes* to confirm.
6. Select *File | Open* and browse to the "Hands-on/Morphology/Solution"
7. Click the file named "HB-Workshop__Flow.cmcards" and press *Open*.
8. Select *Display | Display Options*, and make sure the "Contours" and "Vectors" options are checked.

Create a morphology change dataset

In addition to the "Morphology" dataset that is output from CMS-Flow, it may be desired to see the total change in morphology over time from the initial state. This section of the tutorial will show how to create this dataset.

8. Select *Data | Dataset Toolbox* from the pull-down menu.
9. Click "Data Calculator" under the "Math" section of the "Tools" window.
10. Click the morphology dataset shown under the "Data sets" window.

11. Check the “Use all time steps” check box under “Time Steps”, then click “Add to Expression”. You should see something similar to “h:all” appear in the Calculator window.
12. Click the minus button
13. Uncheck the “Use all time steps” check box under “Time Steps”, then select the zero time (21153 00:00:00), and click “Add to Expression”.
14. The final expression in the calculator should be something similar to “h:all – h:1”. The letter may differ, but the rest should be the same.
15. Give a name for the resulting dataset, such as “Bed Change” in the “Output data set name” field
16. Press *Compute* and wait for the process to complete. A status is red is shown on the bottom-most line of the SMS main window.
17. Press *Done* and verify that the new dataset has been added to the data tree.

Use the techniques defined in the “Viewing contour” section as shown in the Hydrodynamics tutorial to change the appearance of the “Bed Change” dataset contours. We will use the following settings:

6. Contour Method: Color Fill using Color Ramp with the following options
    a. Select “User Defined”
    b. Click “New Palette”, then from the resulting dialog, choose “Magnitude Difference”. NOTE: The magnitude difference palette is best used with a symmetric range to give exactly 0.00 in the middle (ie. -2.0 to 2.0).
    c. Click “Reverse” button to reverse the colors. Typically when showing velocity bathymetry difference, negative values (accretion) are designated with warmer colors and positive values (erosion) are designated with cooler colors. This palette has a neutral color (white) in the middle.
    d. Click *OK* twice,
7. Specify a range: **Checked** and enter values of -7.0 for minimum and 7.0 for maximum.
8. Specify precision: **Checked** and enter value of 3
9. Contour Interval: “Number of Contours” – 9 (or some odd value so that the middle number is exactly 0.0). this will ensure that the areas of no change are white.
10. Legend: **Checked** and for the options:
    a. Title: Bed Change, m
    b. Height 200 pixels and Width 20 pixels
11. Leave the remaining options with their default values.

The resulting screen will appear similar to what is shown in Figure 45.

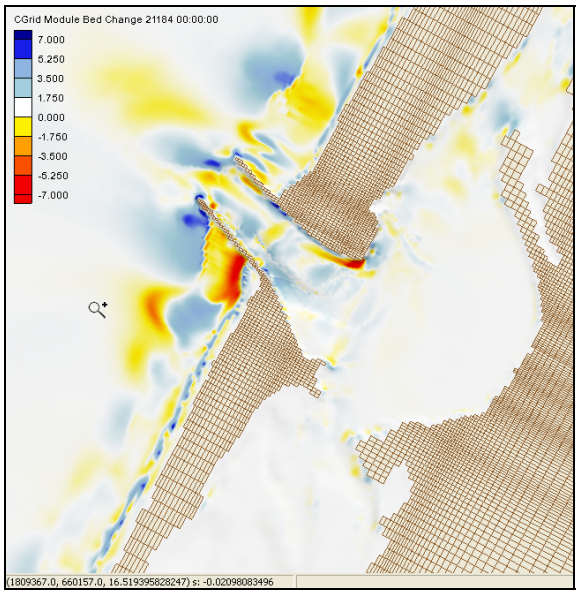


Figure 45. Bed change contour display

This concludes the morphology section of the tutorial.

14.b.3. Salinity Hands-On: Humboldt Bay, CA

Introduction

Salinity refers to the salt content of water. Its value runs from 0 for fresh water to 31-35 ppt (parts per thousand) for ocean water. However, for water bodies with limited water exchange and experiencing evaporation, salinity can run higher and lead to formation of brine. The table, taken from the Wikipedia, gives typical values and nomenclature:

Water salinity			
Fresh water	Brackish water	Saline water	Brine
< 0.05 %	0.05 – 3 %	3 – 5 %	> 5 %
< 0.5 ppt	0.5 – 30 ppt	30 – 50 ppt	> 50 ppt

Salinity can change concentrations of fine-grained sediment and in part defines water quality in coastal and estuarine systems. Salinity changes in time and space, and it cannot only alter estuarine circulation patterns, but also sediment transport.

Presently, CMS-Flow can perform 2-D salinity computations. Validity of 2D salinity implies strong mixing through the water column, such as in estuaries that experience strong wind. Development for the 3-D model is underway. To calculate salinity transport, you will need the regular hydrodynamic input files (Figure 46),

- HB-Workshop__Flow.cmcards,
- HB-Workshop__Flow_grid.h5,
- HB_Workshop__Flow_mp.h5,

and you will also need to prepare initial salinity conditions and salinity boundary conditions. You can locate the files in the “Hands-on/Salinity” directory.

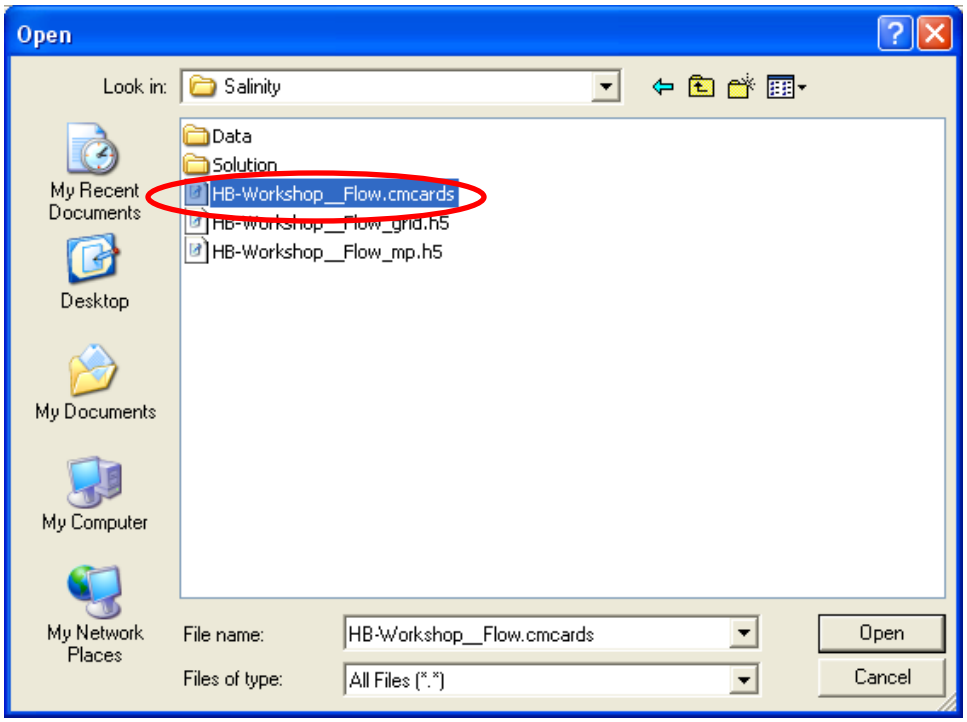


Figure 46. Opening the files for the CMS-Flow simulation with salinity.

Initial Salinity Conditions

Salinity transport is computed under CMS-Flow. Most of the setup will be completed in the *Cartesian Grid* module.

- Constant initial salinity

    1. Open the CMS-Flow file, HB-Workshop__Flow.cmcards (Figure 46),
    2. Choose *CMS-Flow | Model Control*,
    3. Click on the *Transport* and turn on the *Calculate salinity*,
    4. Set the *Global concentration* to 33.5 ppt,
    5. Set the *Transport rate* under the *Time steps* to calculate salinity field (Figure 47).

- Spatially-varying initial salinity (under development)

    1. Generate a sampling salinity file (salt_init.xyz),
    2. Run the Laplace interpolation program to generate a 2-D salinity field.

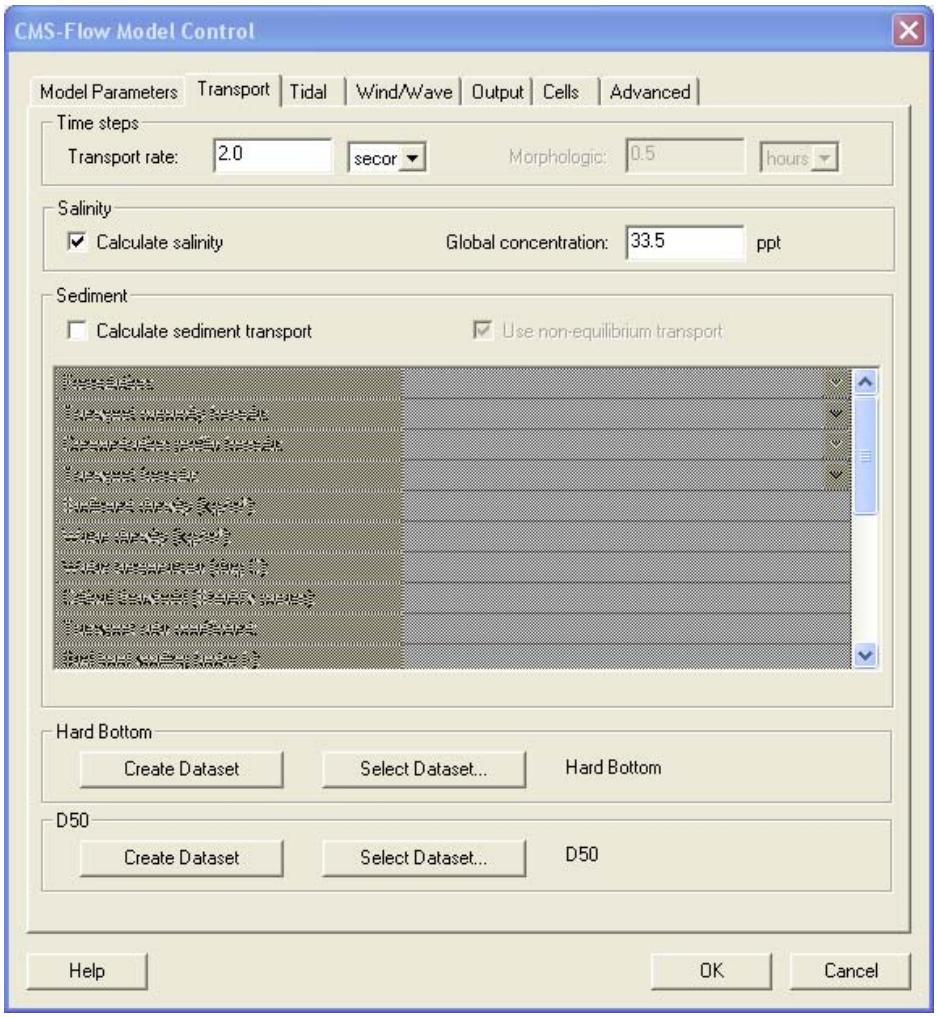


Figure 47. Setting up the salinity calculation.

Salinity Boundary Conditions

Salinity values need to be specified at model boundaries for numerical calculations.

- WSE (Water Surface Elevation) boundary

    1. Select the *WSE-forcing* boundary cellstring by choosing the *Select Cellstring* tool,
    2. Choose *CMS-Flow | Assign BC*,
    3. Click the *Curve undefined* under *Salinity* on the left hand side of the dialog (Figure 48),

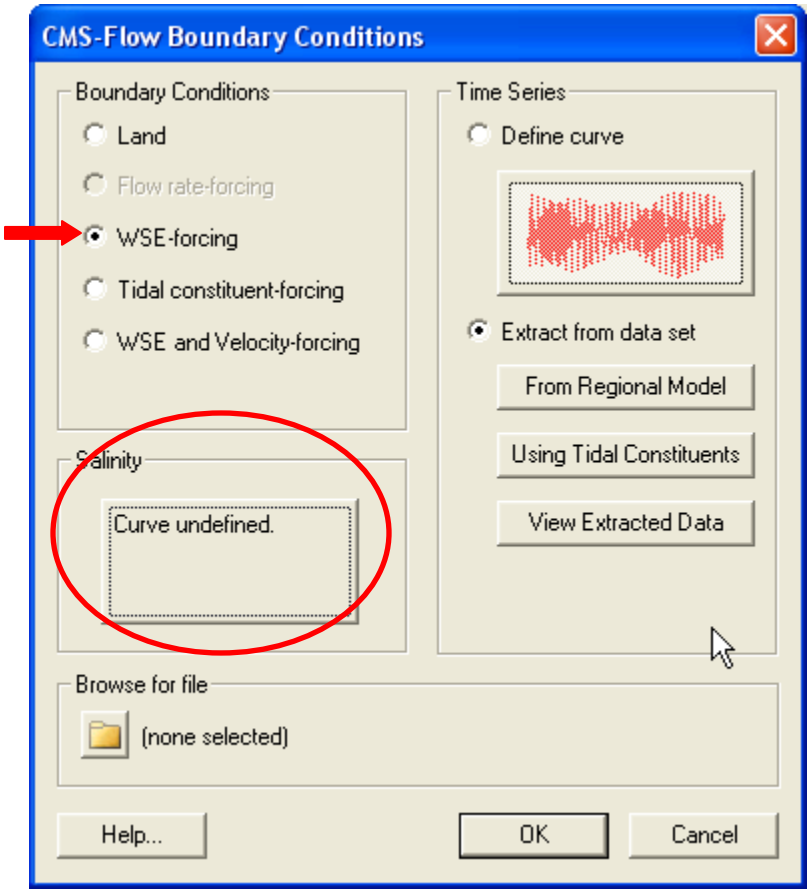


Figure 48. Salinity specifications along the *WSE-forcing* boundary.

4. Click the *Import* button to read time series of salinity along the boundary in xys format, Trinidad.xys (Figure 49),

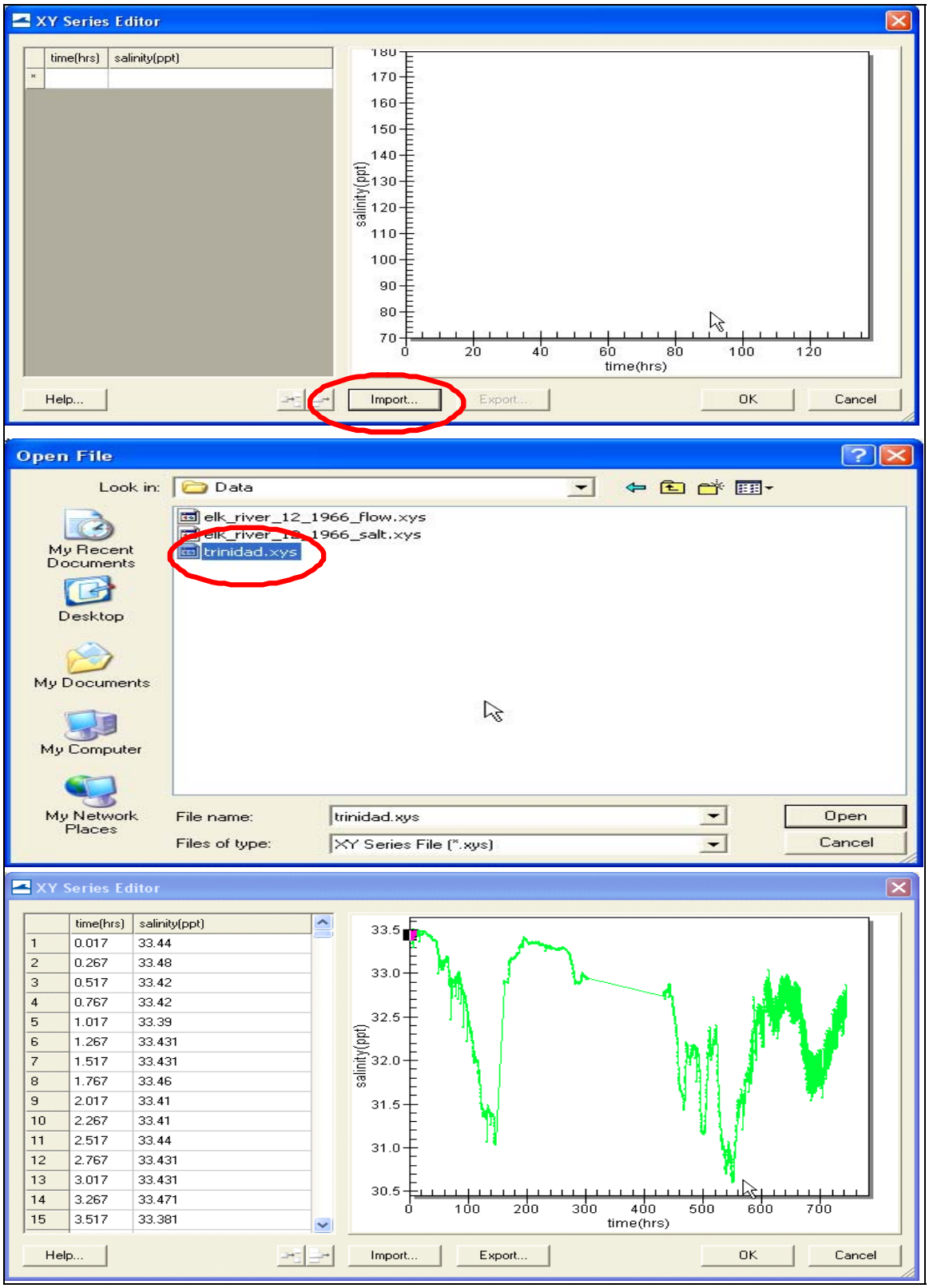


Figure 49. Salinity data from a xys file.

5. Click *OK* to exit *XY Series Editor*,
6. Click *OK* to complete salinity setup along the *WSE-forcing* boundary.

- River boundary

5. Select the *Flow rate-forcing* boundary cellstring by choosing the *Select Cellstring* tool,
6. Choose *CMS-Flow | Assign BC*,
7. Click the *Curve undefined* under *Salinity* on the left hand side of the dialog (Figure 50),

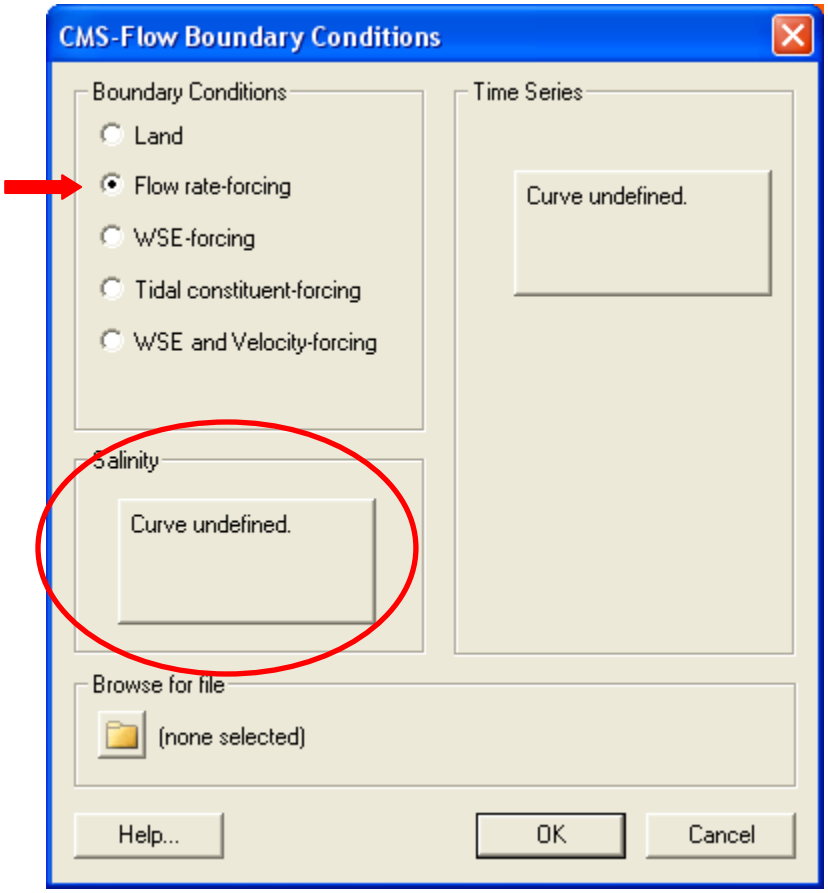


Figure 50. Salinity specifications along the *Flow rate-forcing* boundary.

8. Click the *Import* button to read time series of salinity along the boundary in xys format, elk_river_12_1966_salt.xys,
9. Click *OK* to exit *XY Series Editor*,
10. Click *OK* to complete salinity setup along the *Flow rate-forcing* boundary.

Along with the salinity specifications you will also need to specify river flows. As shown in *XY Series Editor*, salinity boundary values and flow rates can always be imported manually or from an opened *Excel* data file using *Copy/Paste*.

Salinity Output

Once selecting the salinity calculations, you will need to update the *Output* options in *CMS-Flow | Model Control* for your solution file.

Viewing the Salinity Field

A CMS-Flow simulation with salinity transport will generate an extra scalar data subsets within the hydrodynamic solution file, *_salinity, in a typical CMS-Flow solution data tree (Figure 51).

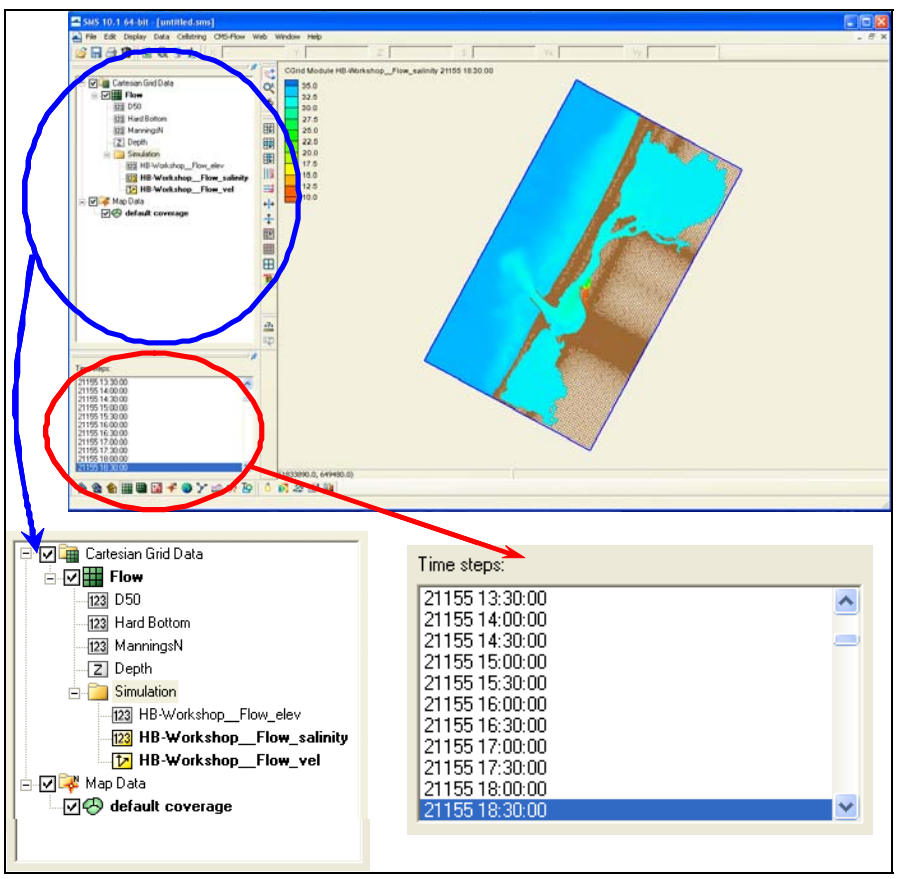


Figure 51. CMS-Flow simulation with salinity in Humboldt Bay.

- Loading the solution file

    1. Choose *File | Open,*

2. Browse your folders to find the solution file, HB-Workshop__Flow_sol.h5, in the “Hands-on/Salinity/Solution” directory.
3. Press *Open*.

- Display salinity distribution (Figure 52)

1. Click/highlight the salinity output, HB-Workshop__Flow_salinity, under the *Simulation* in the *Cartesian Grid Data* tree,
2. Choose *Display | Display Options*,
3. Click *Cartesian Grid* on the left hand side of the dialog,
4. Check the *Contours* on the right hand side of the dialog,
5. Select the *Contours* button at the top of dialog,
6. Choose the *Contour Method | Color Fill*,
7. Click the *Color Ramp* button to adjust the color scheme,
8. Set the proper salinity range by checking the *Specify a range* (10-35 ppt),
9. Change the *Contour Interval* to 11, plotting salinity contours every 2.5 ppt,
10. Click *OK* to exit *Display Options*.

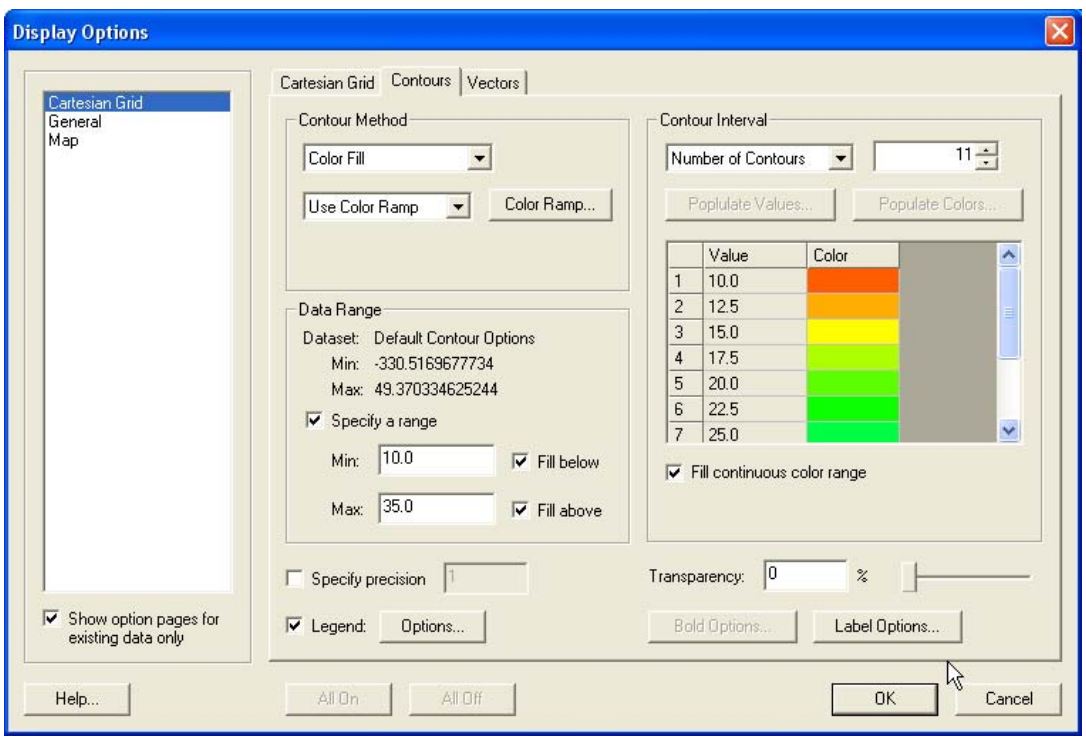


Figure 52. Setting up salinity display in the *Display Options*.

At the low left corner of *SMS 10.1* window, there is the *Time steps* box (Figure 51). User-specified output time steps (30 min in this example) are shown in the box. Click/highlight

each time step, the corresponding simulation results are displayed on the right hand side of the window (Figure 51).

14c. Setting Up CMS-Wave[1]

1. Grid Transformation

The standard method to generate a CMS-Wave grid is the same as to generate the CMS-Flow grid in the *Cartesian Grid* module (make sure you are in the CMS-Wave mode). The only step that needs special attention is to start the Cartesian grid generation from the grid origin location (the 1st point to click in the grid generation process). The x axis (positive direction) needs to point towards the shoreline and y axis needs to be parallel to the shoreline.

Alternatively, a CMS-Wave grid can be transformed from a CMS-Flow grid if the Flow grid has already been generated. For example, we can transform a Humboldt Bay existing CMS-Flow grid, Flow_HB.cmcards, to a CMS-Wave grid (the new CMS-Wave grid will look exactly the same as the CMS-Flow grid). If a smaller CMS-Wave grid domain (smaller than the CMS-Wave grid, say, if some bayside or landside area is not of interest), users can use the SMS *Select Grid Row* or *Select Grid Column* tool to merge cells including boundary cells and use *Drag Row Bound*ary or *Drag Column Boundary* to adjust the new boundary cell size (may need to remap the correct depth and elevation from the scatter data to the new boundary cells).

3. Open the CMS-Flow file, Flow_HB.cmcards,

4. Choose *Data | Switch Current Model*,

5. Select *CMS-Wave* and click *OK*,

6. Check the transformed CMS-Wave grid origin for the correct grid orientation (the correct origin shall have the corner cell with i=1 and j=1)

7. If the transformed CMS-Wave has the correct origin, congratulations, you now have the CMS-Wave grid that is ready for use – save your CMS-Wave grid files (save under the simulation *.sim filename for all wave input files or under the SMS project *.sms filename for the complete project files including *.sim). If not, follow Steps 6 and 7 to get the correct CMS-Wave grid orientation.

8. Right click 'Flow' in the data-tree directory under *Cartesian Grid Data* and choose *Create Transformed Grid*.

[1] Setting up CMS-Wave sample files are located in the "Hands-on/Setting-up-CMS-Wave" directory.

9. Select the correct *Rotation* angle (deg), choose from 90, 180, and 270 deg, to relocate the CMS-Wave origin for the correct grid orientation. Save the CMS-Wave *.sim file after setting the final grid orientation.

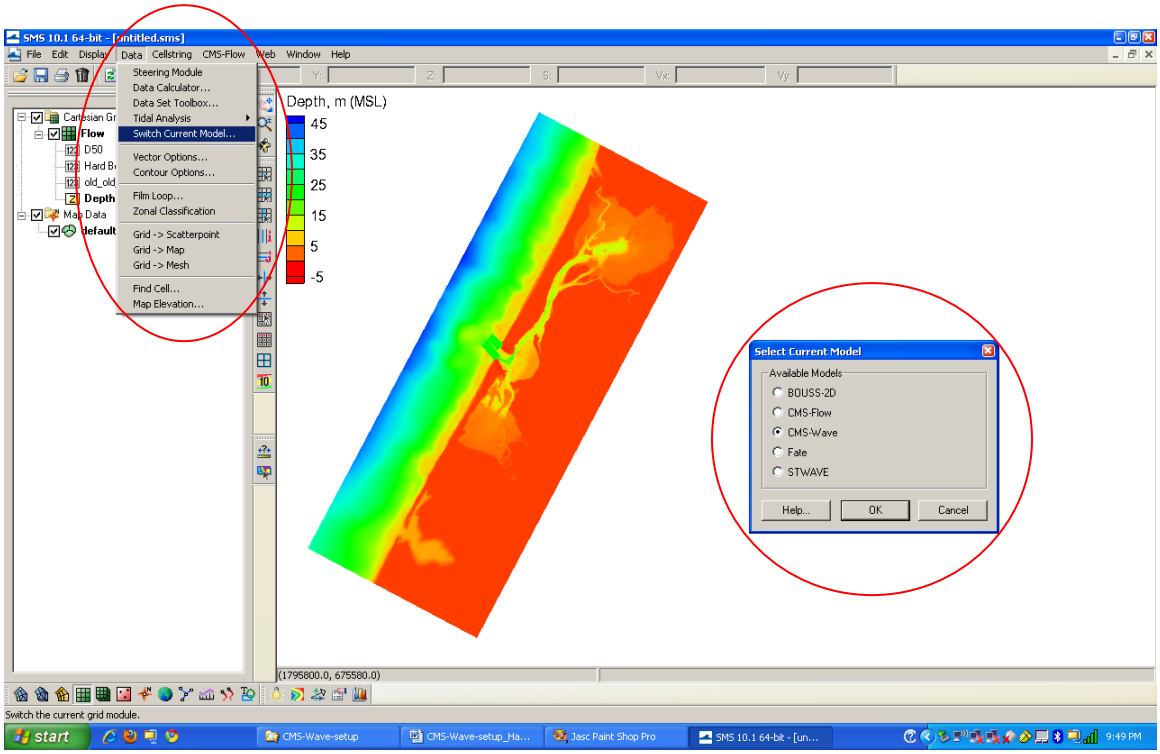


Figure 53. Use *Data | Switch Current Model* to convert CMS-Flow grid to CMS-Wave grid.

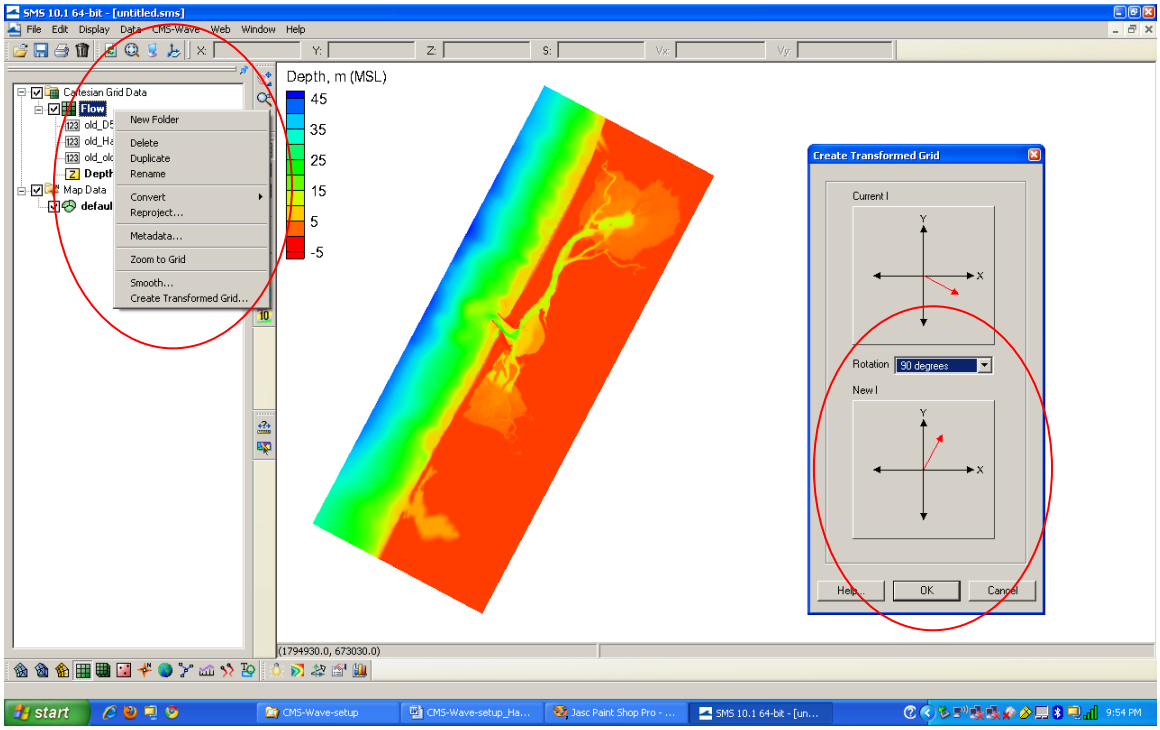


Figure 54. Re-select the grid origin for the correct grid orientation.

2. Input Wave Spectra

Users can use SMS to generate input spectra by specifying the spectral type (e.g., JONSWAP, TMA, and Ochi-Hubble) with associated spectral parameters. For the Humboldt Bay example in Section 1:

1. Open the CMS-Wave file, Wave_HB.sim,

2. Choose *CMS-Wave | Spectral Energy*,

3. Click *Create Grid* and *OK*,

4. Click *Generate Spectra* and select the preferred *Generation Method* (default is TMA shallow water spectrum),

5. Specify *Gauge Depth* (average depth at the seaward boundary) and wave approaching direction *Projection* (default Shore Normal, Oceanographic, or Meteorological),

6. Input *Spectral Parameters* on key board – for example, input a sequence of wave index label, wave angle (deg), wave height (m), wave period (sec), peak enhancement gamma, and directional spreading nn (cosine power function)

7. After input all spectral parameters (one or multiple input spectra), click *Generate* to finish. The generated individual spectrum can be viewed by the click on the corresponding wave index. Click *Done* to exit the *Spectral Energy* mode. Remember to save the *.sim file, this will create the *.eng ascii file that can associated with additional wind and water level input information.

8. After the wave input spectra are generated, uses can choose *CMS-Wave | Model control* to specify wind vector and water level input information,

9. Under *Wave Source*, select *Spectra and wind*, and click *Parameters*,

10. Choose wind *Angle convention* (default Shore Normal, Oceanographic, or Meteorological) if wind input is to be specified (screen input or copy/paste from Excel file),

11. Specify wind and tidal (water level) elevation data and then click *OK* to complete the wind and water level input,

12. Save wave *.sim file.

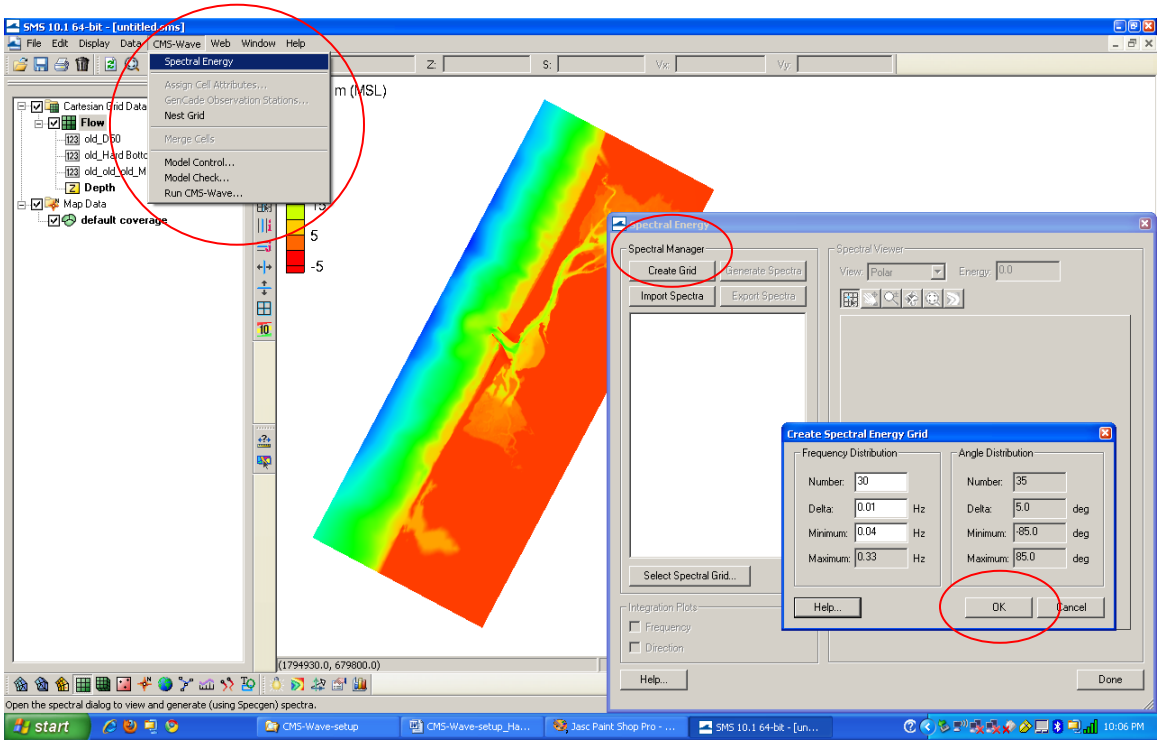


Figure 55. Use *CMS-Wave | Spectral Energy* to create directional spectrum grid.

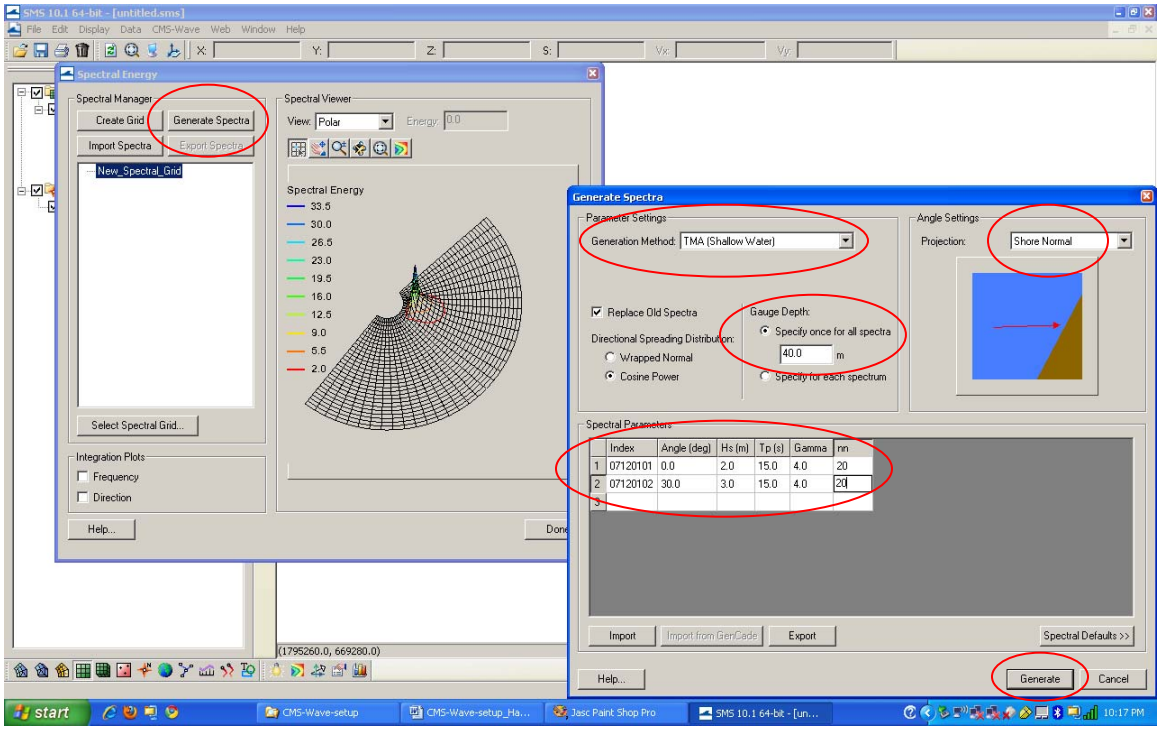


Figure 56. Use *Generate Spectra* and *Generation Method* to input the spectral parameters.

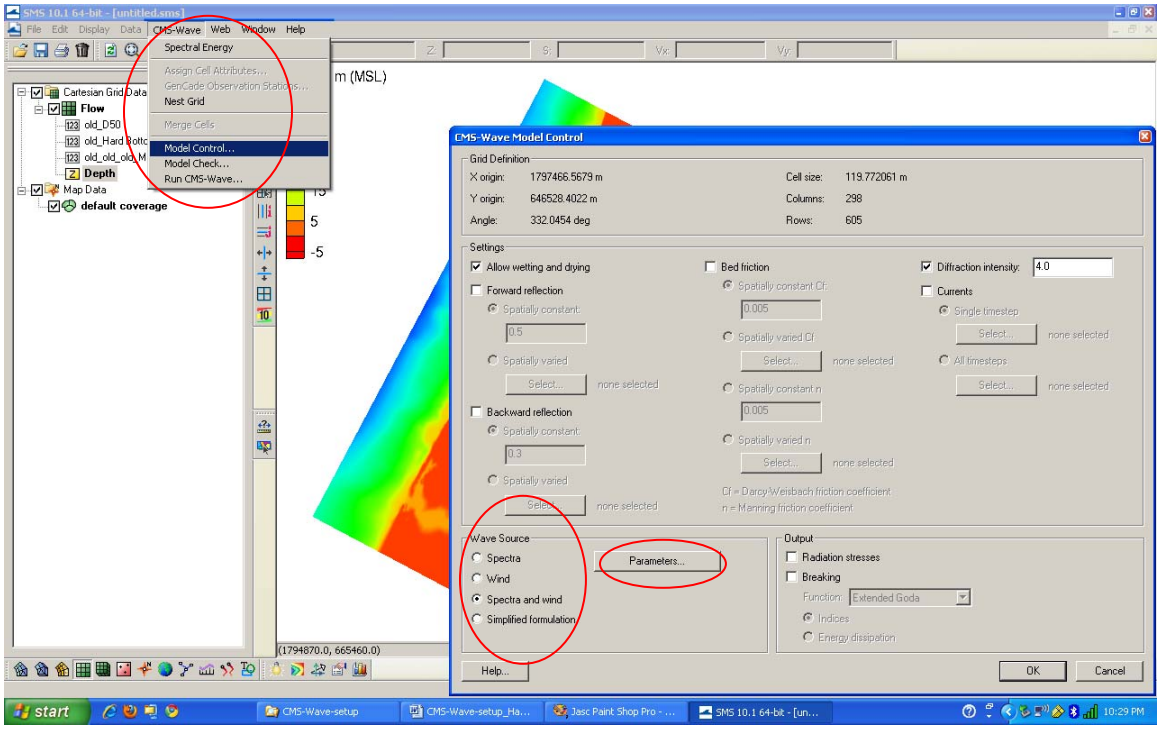


Figure 57: Use *CMS-Wave* | *Model control* and *Wave Source* to provide wind input data.

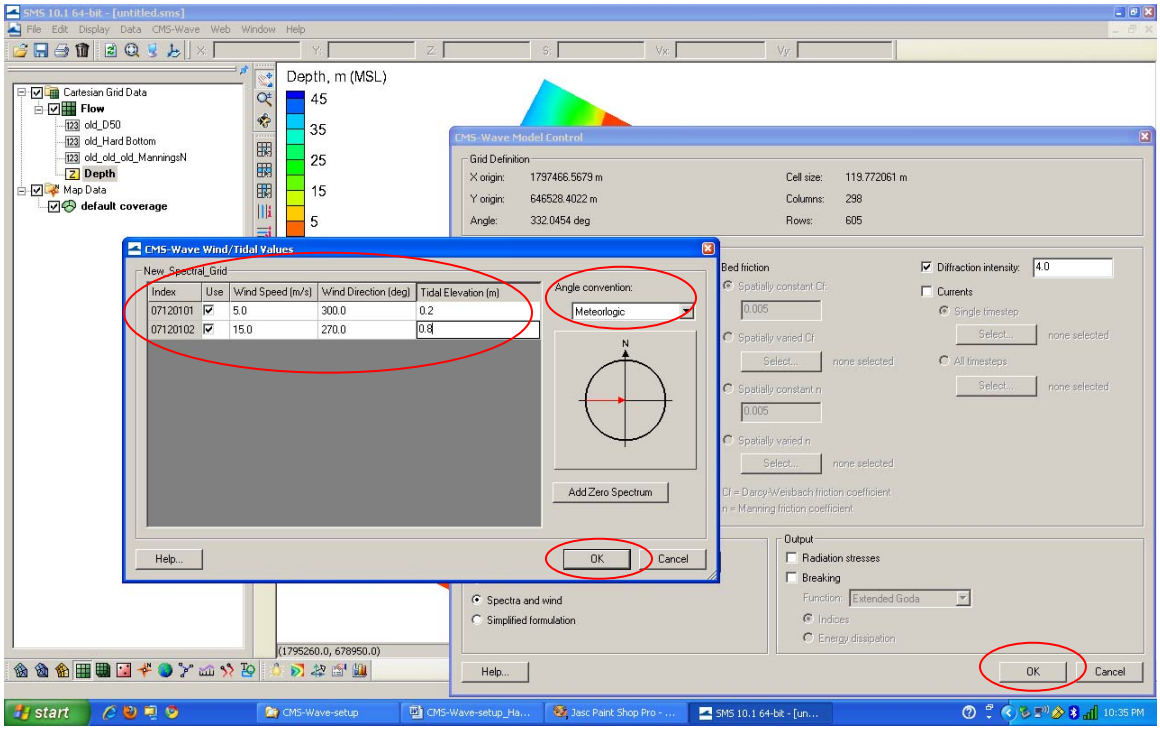


Figure 58. Specify wind input (choose wind angle convention) and water level elevation data.

Directional wave data collected by NDBC or CDIP buoys can be processed for input spectra to CMS-Wave.

- NDBC buoy data – run ndbc-spectra.exe to read the NDBC standard directional wave file and prepare the CMS-Wave input spectral *.eng file.

    1. Download the NDBC standard monthly directional wave spectral file from http://www.nodc.noaa.gov/BUOY/buoy.html (e.g., 46022_200712).

    2. In the DOS mode, run ndbc-spectra.exe,

    3. Responding to the on-screen input, type the NDBC spectral filename,

    4. Type the starting time stamp (default value is 0) for saving output file(s),

    5. Type ending time stamp (default is 99999999) for saving output file(s),

    6. Type the time interval (hr) for saving output data,

    7. Type 2 to save the CMS-Wave *.eng and *.txt files,

    8. Type the local shoreline orientation (actually the CMS-Wave grid y axis) in Polar Coordinates (deg, positive from North covering the sea, e.g., 180 deg for St Mary's Entrance, FL/GA),

9. Type the NDBC buoy location water depth (m) and then the CMS-Wave seaward boundary mean water depth (m),

10. Type 1 to include wind or 0 to skip the wind, and to complete the run,

11. Rename fort.15 to *.txt and fort.18 to *.eng (half-plane wave energy),

12. Two additional files are generated: fort.12 and fort.19 tabulate the wave (and wind) parameters for the full-plane and half-plane wave propagation coverage, respectively (Matlab fig1set.m and fig2sets.m can read these files to plot the wave and wind data time series).

- CDIP buoy data - run cdip-spectra.exe to read the CDIP standard directional wave file and prepare the CMS-Wave input *.eng file. Download the CDIP wave file from http://cdip.ucsd.edu/?nav=historic&sub=data (e.g., sp128-200712).

  Follow the steps as processing the NDBC buoy data – the only difference is that the CDIP wave spectra processed are used directly as input to CMS-Wave (so no need to transform the CDIP wave spectra to CMS-Wave seaward boundary).

- Examples are CDIP 128 and NDBC 46022 standard spectral files for Dec 2007.

3. Model Parameters

Saving the *.sim file in SMS will also produce *.dep (depth info), *.std (model parameters), and *.struct (feature cells) files, where only *.sim and *.dep are required when open *.sim in SMS. To run CMS-Wave, only four input files *.sim, *.std, *.dep, and *.eng are required. CMS-Wave parameters can be specified in SMS under the *CMS-Wave | Model control*. However, several new parameters have not yet included in the present SMS version and need to be specified manually in the *.std file.

Additional input files including *.cur (current field input), *.eta (water surface field input), *.txt (summary of input wave parameters), *.struct , friction.dat (bottom friction coefs field), forward.dat (forward reflection coefs field), backward (backward reflection coefs field), and mud.dat (mud-bed turbulent viscosity coef field) are optional.

The *.std file can look like (Wave_HB.std)

```
0  0  0  0  2  0  0  3  0  0  4.0  0.0250  0.500  0.300  0  1  0  0  2  1
53   292
202 244
```

The first row shows the model parameters – the first six parameters are required, the rest (up to a total of 20 parameters) are optional. The optional second and more rows are the

special output location (x and y) indices for wave monitoring stations and for the child grid wave input stations.

The 20 model parameters are (in sequential order)

iprp icur ibrk irs kout ibnd iwet ibf iark iarkr akap bf ark arkr iwvbk nonln igrav isolv ixmdf iproc

which represent:

iprp = 0, for wave generation and propagation (use wind input if provided)

= 1, for propagation only (neglect wind input)

= -1, for fast-mode simulation (for wave generation and propagation)

icur = 0, no current

= 1, with current input (*.cur), using data sets in the sequential order

= 2, with current input (*.cur), using only the first set current data

ibrk = 0 (no *.brk file)

= 1, for output of wave breaking indices (*.brk)

= 2, for output of energy dissipation fluxes (*.brk)

irs = 0 (no *.rad file)

= 1, for output of wave radiation stresses (*.rad)

= 2, for output of wave radiation stresses (*.rad) and wave setup/maximum water level (setup.wav)

= 3, for automatic wave run-up calculation

kout = 0 (no *.obs and selhts.out files)

= n, for output of spectra (*.obs) and parameters (selhts.out) at n selected cells

ibnd = 0 (no *.nst file)

= 1, for nested grid, with linear interpolation of boundary input spectra (*.nst)

= 2, for nested grid, with morphic interpolation of boundary input spectra (*.nst)

iwet = 0, for normal wetting/drying (use water level input)

= 1, no wetting/drying (neglect water level input)

ibf = 0, no bottom friction

= 1, for bottom friction with constant Darcy-Weisbach type coefficient (= bf)

= 2, for bottom friction with variable Darcy-Weisbach type coefficient (friction.dat)

= 3, for bottom friction with constant Manning coefficient (= bf)

= 4, for bottom friction with variable Manning coefficient (friction.dat)

iark = 0, no forward reflection

= 1, with forward reflection

iarkr = 0, no backward reflection

= 1, for backward reflection

akap = diffraction intensity factor (0 for no diffraction, 4 for strong diffraction)

bf = constant bottom friction coefficient

ark = constant forward reflection coefficient (0 for no reflection, 1 for maximum forward reflection)

arkr = constant backward reflection coefficient (0 for no reflection, 1 for maximum backward reflection)

iwvbk = option for selection of wave breaking formula

= 0, for Extended Goda (Sakai et al. 1989)

= 1, for Extended Miche (Battjes 1972; Mase et al. 2005b)

= 2, for formula by Battjes and Janssen (1978)

= 3, for formula by Chawla and Kirby (2002)

nonln = 0, no nonlinear wave-wave interaction

= 1, with nonlinear wave-wave interaction

igrav = 0, no infra-gravity wave effect

= 1, with infra-gravity wave effect

isolv = 0, for alternative direction iterative solver

= 1, for high-speed Gauss-Seidel solver with multi-processor capability

= 2, for standard Gauss-Seidel solver

ixmdf = 0, for input/output in ASCII format

= 1, for input/output in XMDF format

iproc = 0, with a single-processor (default)

= n, with n processors (only valid with isolv = 1)

4. Structure Types

CMS-Wave calculates wave transmission, wave runup, and overtopping as special features on selected cells. These cells can represent jetties, floating breakwaters, a bottom mound breakwater, or underwater features such as reefs or submerged structures. A trench or submerged mound can be added to the bed as features without modifying the input depth file. These feature cells are specified in the *.struct file. Each feature cell is described by four parameters, istruc, jstruc, kstruc, and cstruc in the *.struct file.

istruc = i-th column in the grid

jstruc = j-th row in the grid

kstruc = feature cell identity

= 1, for adding alternative feature or structure (immersed or exposed) without modifying the input depth

= 2, for calculation of wave runup and overwash on beach face or structure, and adjacent land

= 3, for calculation of transmitted waves of a floating breakwater

= 4, for a vertical wall breakwater

= 5, for a composite or rubble-mound breakwater

= 6, for high porous breakwater

= 7, for low porous breakwater

cstruc = feature structure depth, for kstruc = 1 (assume a land cell if not provided)

= beach/structure elevation above mean water level, for kstruc = 2 (use the input depth if not provided; no effect for cstruc < 0)

= floating breakwater draft, for kstruc =3 (skip if not provided or cstruc < 0.05 m)

= breakwater/structure elevation, for kstruc = 4 or 5 (use the input depth if not provided; immersed if cstruc < 0)

= ambient water depth for kstruc = 6 or 7

Users can specify feature cells in SMS. For the Humboldt Bay example in Section 1:

1. Open the CMS-Wave file, Wave_HB.sim,

2. Click *Select grid cell* tool and proceed to select cells (use the polygon or click a cell while holding the Shift key or drag the mouth while holding the Ctrl key),

3. Choose *CMS-Wave | Assign Cell Attributes*,

4. Select *Structure* and choose *Type* (rubble-mound breakwater, wave run-up, etc.) for the proper cell features,

5. Select *Use modification* for any new elevation or additional depth (or characteristic length quantity) and *OK* to exit.

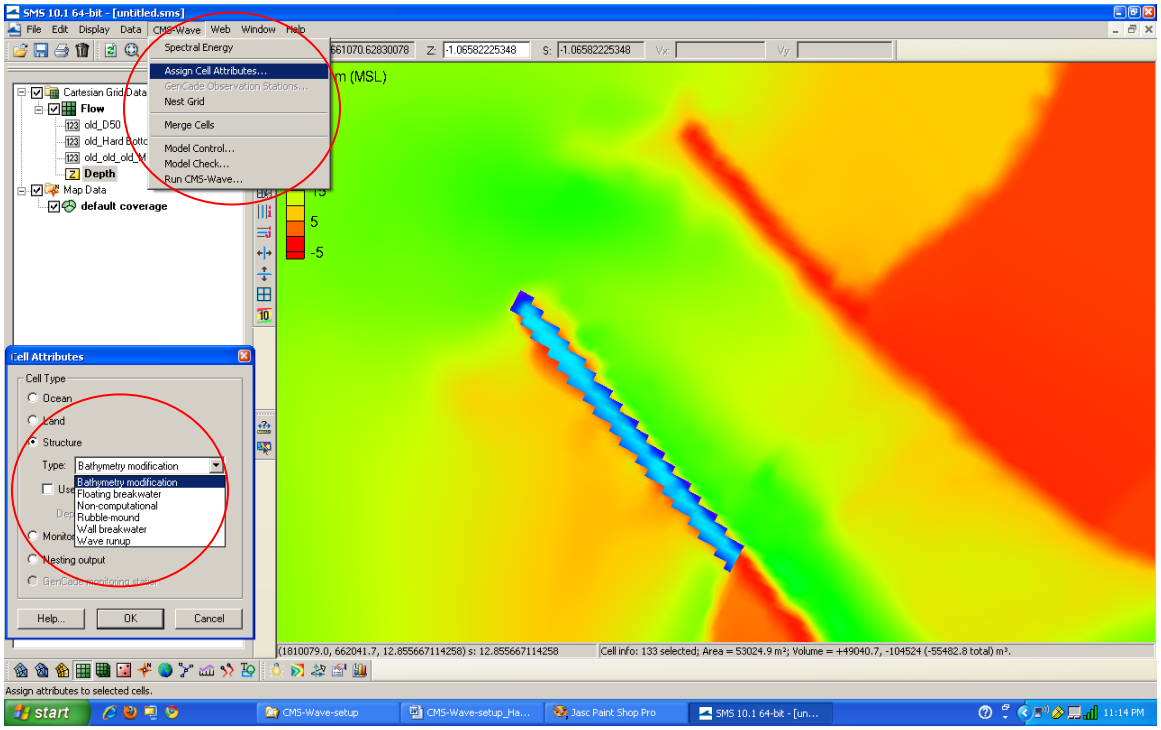


Figure 59. Use CMS-Wave | Assign Cell Attributes to specify feature cells (structure type).

5. Wave Run-up and Overtopping

Wave run-up is triggered by either providing the Type 2 feature cells in the *.struct file or setting the model parameter irs = 3 (will automatically trigger the wave run-up for the entire grid domain). The overtopping rate is calculated and reported at the special output (wave monitoring station) locations. The wave setup and maximum water level fields (including wave run-up) are saved in the setup.wav file.

Guide to interaction between CMS-Wave and CMS-Flow (SMS Steering Module)

1. Select *File | Open* on the pull-down menu and load both the “Flow_HB.cmcards” and the “Wave_HB.sim” files from the “Hands-on/Setting-up_CMS-Wave” directory.
2. Select *Data | Steering Module* from the pull-down menu.
3. Choose selections on “Steering Wizard” window
   a. Run CMS-Wave every ____ hours – This is the frequency that you wish to iterate between CMS-Flow and CMS-Wave. Normally this is governed by your wave characteristic data.
   b. CMS-Flow -> CMS-Wave – This is the information to be passed from CMS-Flow back to the CMS-Wave model upon completion of each iteration.
   c. CMS-Wave -> CMS-Flow – This is the information to be passed from CMS-Wave back to the CMS-Flow model upon completion of each iteration.
   d. Press *Start* to begin the steering process.

This process is shown in Figure 60.

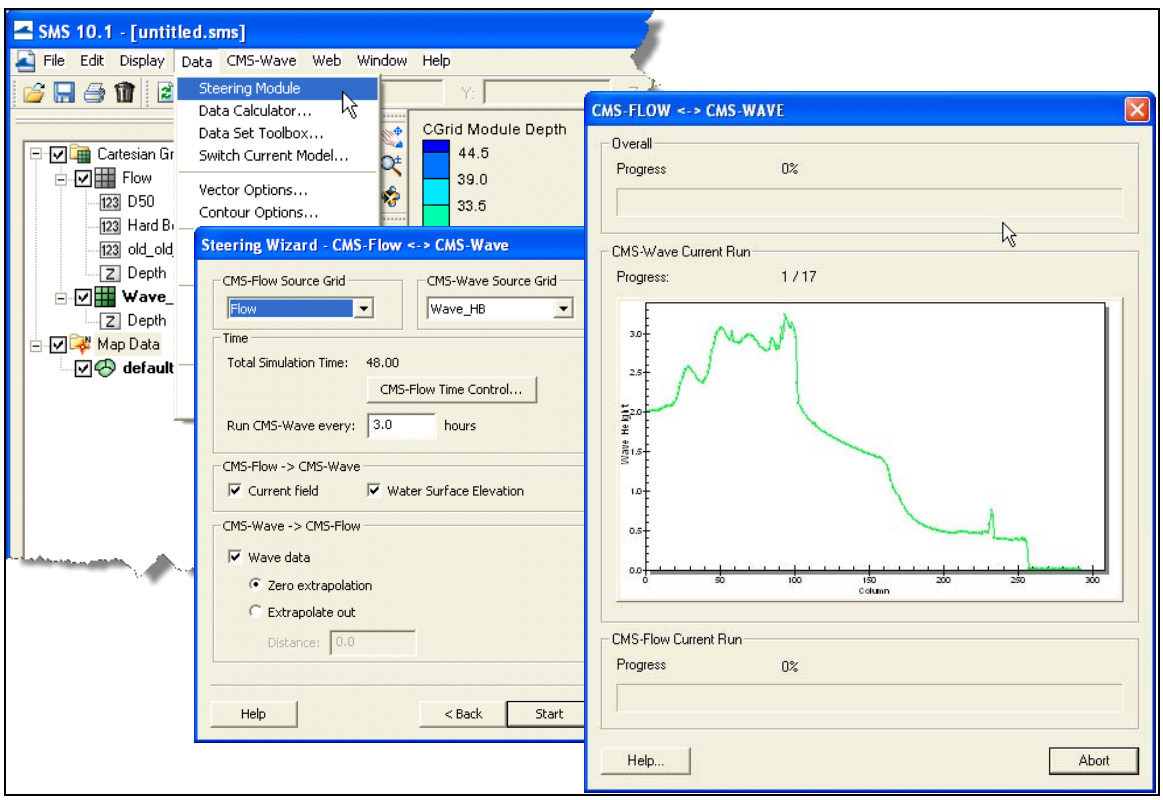


Figure 60. Steering module dialogs

14.d. Calculating Channel Infilling Hands-On Practice: Humboldt, CA

Introduction

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. In the next section (14.e), sediment transport calculations will be extracted from an arc. This section will cover the methodology used to extract time series morphology change data for the purpose of illustrating channel infilling.

Channel Infilling - Time Series Profiles

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.

- Change Depth to Elevation

    1. Load the SMS file run1.sms (and solution file, if it does not load automatically) located in the directory *Hands-on/Humboldt Solutions/Run 1*,

    2. Click *Data*, *Data Calculator*, and the Data Calculator should be selected in the Tools section as shown in Figure 61,

    3. Select the *Bathymetry* scalar (123) dataset, under *Time Steps* check the *Use all time steps* on, and double-click the *Bathymetry* (highlighted in Figure 61),

    4. The line under Calculator will display e:all, add the multiplication symbol (*) and negative 1 (-1) as shown in Figure 61,

    5. Change the *Output data set name* similarly to Figure 61 so as to distinguish this dataset from the original *Bathymetry* output, click *Compute*, and after the scalar set appears in the SMS window (Figure 62), click *Done*.

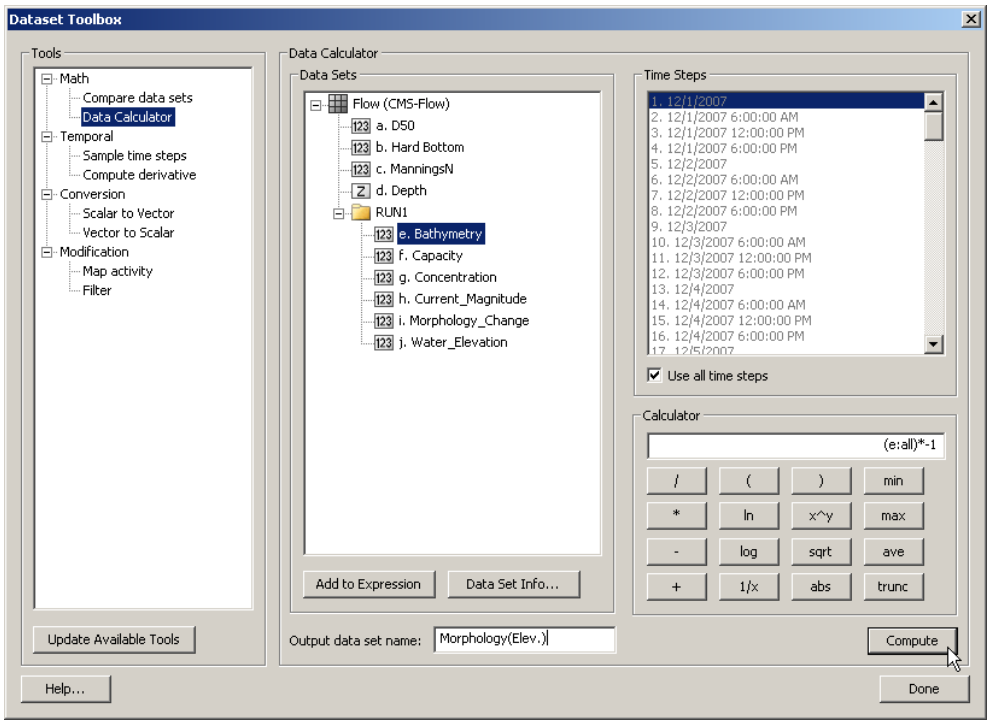


Figure 61. ***Data Calculator Options*** **in the** ***Dataset Toolbox*****.**

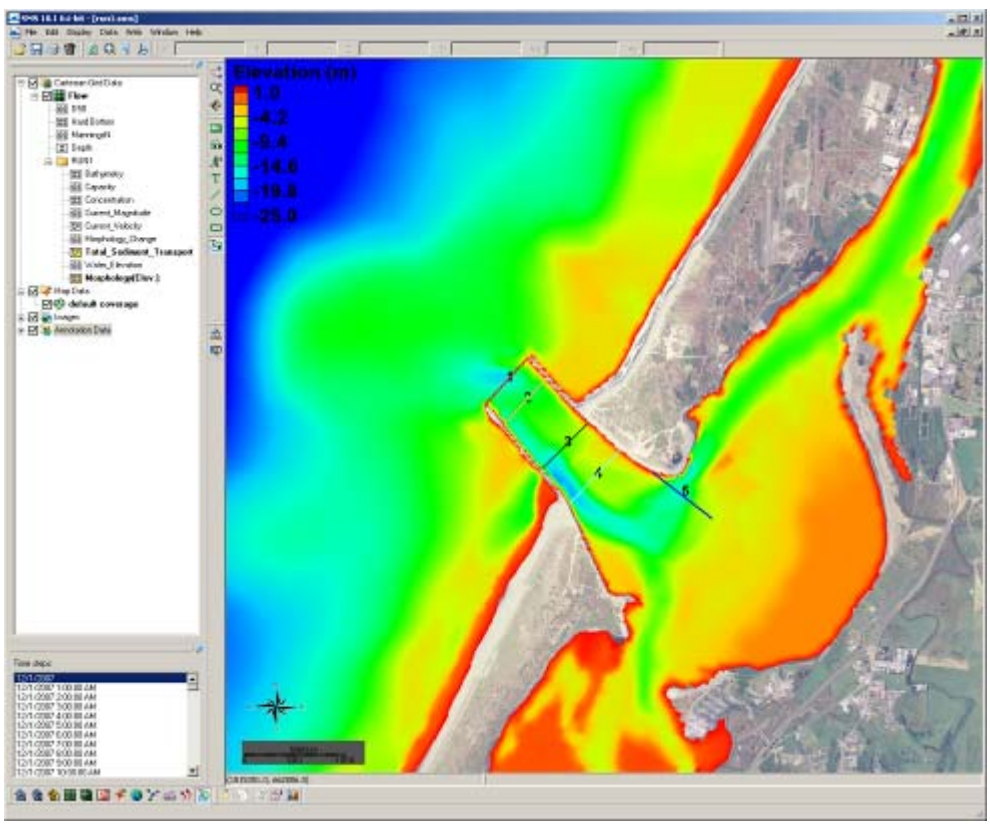


Figure 62. Display of calculated bathymetry now converted to elevations. Note the legend has been modified to display the new contours.

The next step is to generate the cross sections, or arcs, in SMS to extract data with.

- Create Feature Arcs for Extracting Time Series Morphology Change

    1. Click the *Map Module* button , or Map Data

,

    2. Rt-click on *default coverage*, and under *Generic*, *Observation*, select *CMS-Flow*,

    3. Click the *Create a Feature Arc* button to generate the arcs that cross sections will be extracted from, and click on one side of the channel and double-click on the other side to close the arc,

    4. Click the *Select a Feature Arc* button and select the generated arc (Note that there is a direction associated with the arc),

    5. Rt-click and click on *Attributes*, and set the color and name(s) of each arc which are listed together in the *Observation Coverage* options (Figure 63), click *OK*.

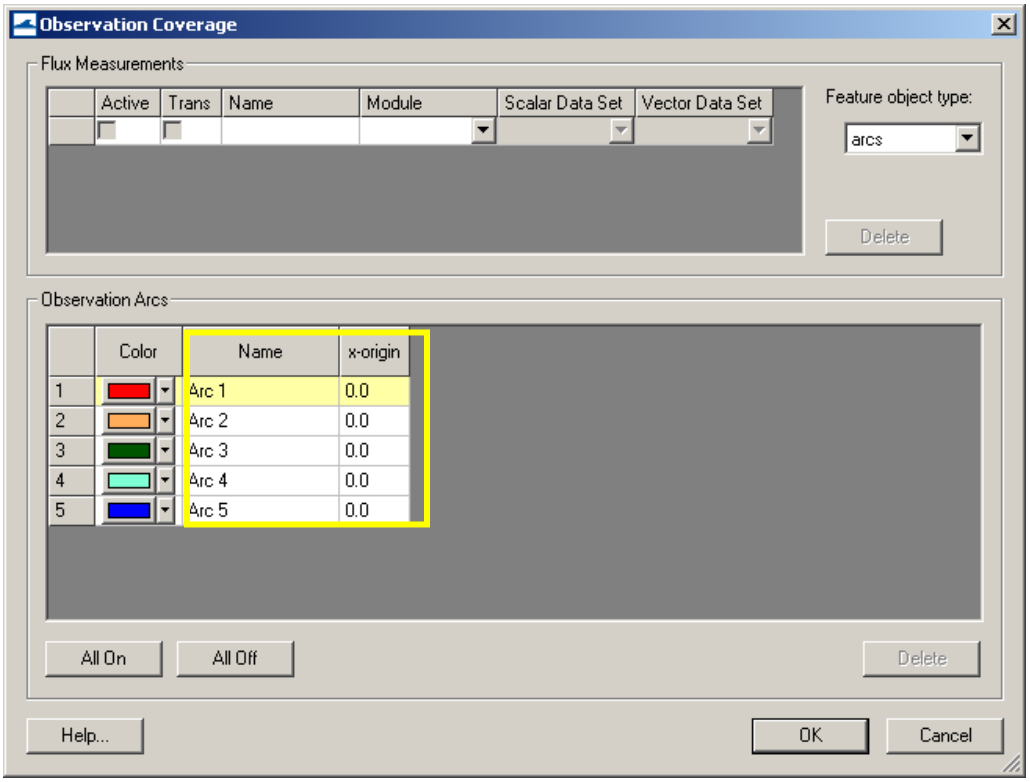


Figure 63. Observation Coverage options.

The following step is the final step for generating the graph in SMS. This is also the present method to extract and export this information in text format for use in other plotting software.

- Plotting Time Series Morphology Change in SMS

    1. Click *Display*, *Plot Wizard*, and click Observation Profile (which should have the display in Figure 64 and *Next* should be available if the arcs were generated in the *Observation Type Coverage* in the above *Create an Arc* section), click *Next*,

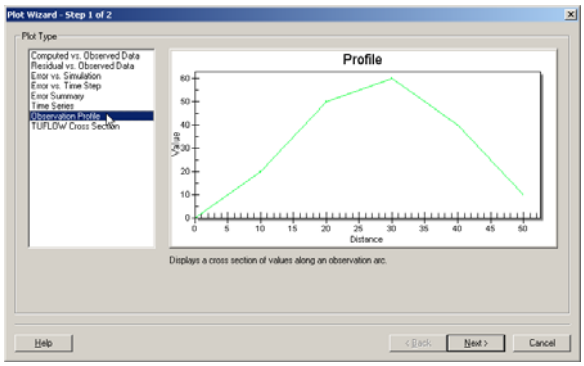


Figure 64. Plot Wizard Step 1.

    2. Following Figure 65, 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*,

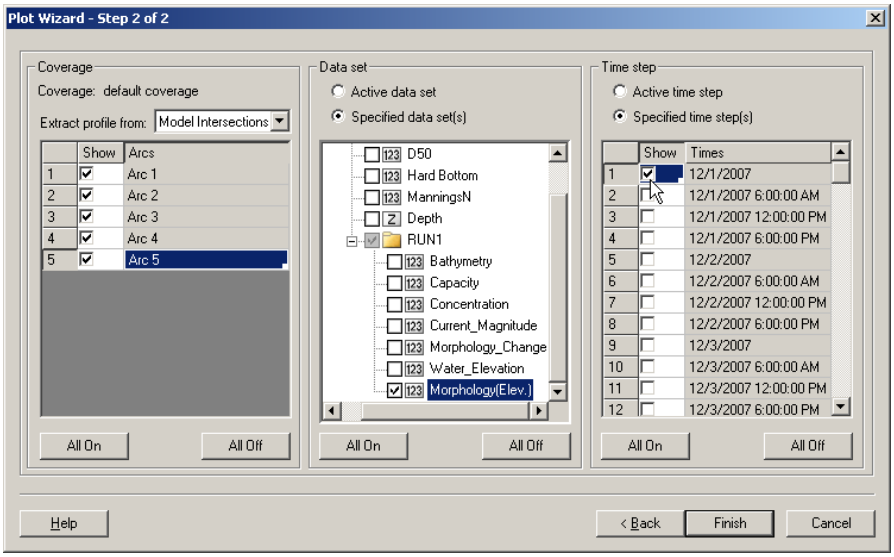


Figure 65. *Plot Wizard* Step 2.

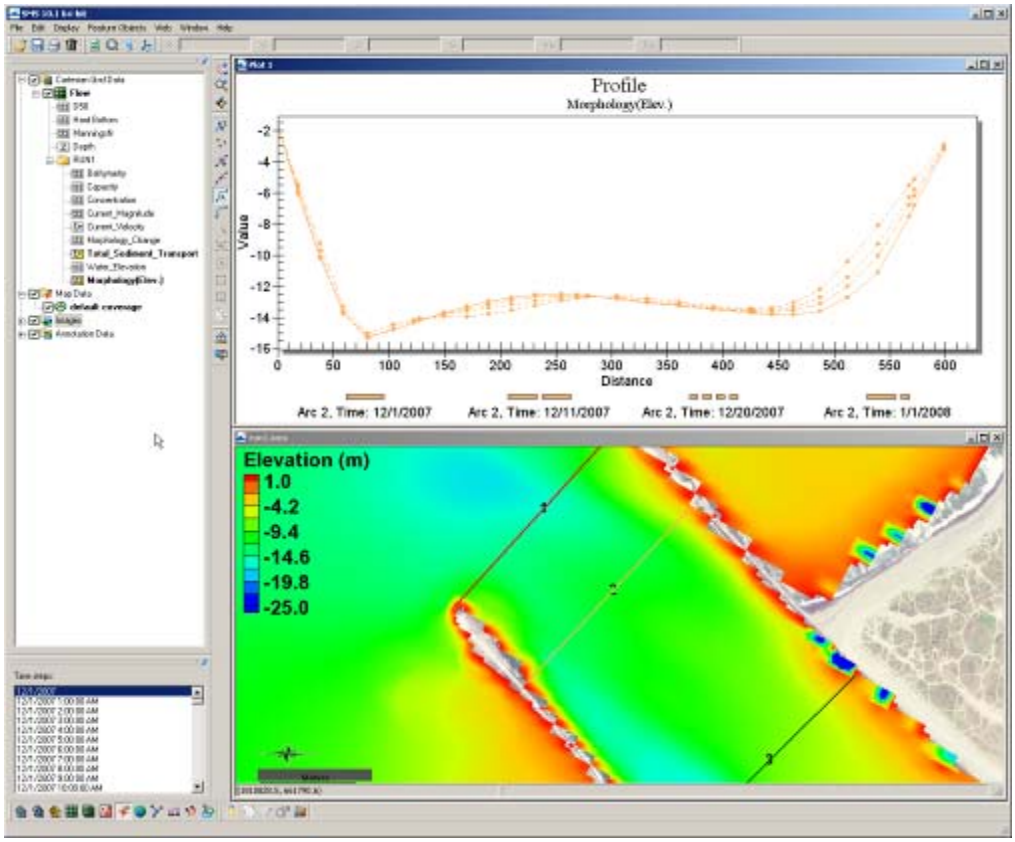


Figure 66. Plotted cross-section.

3. Rt-click on the plot (Figure 66) 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),

4. 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 67),

5. To add more profiles or time-series, rt-click on the plot, and select *Plot data* to bring up the original options from Figure 65.

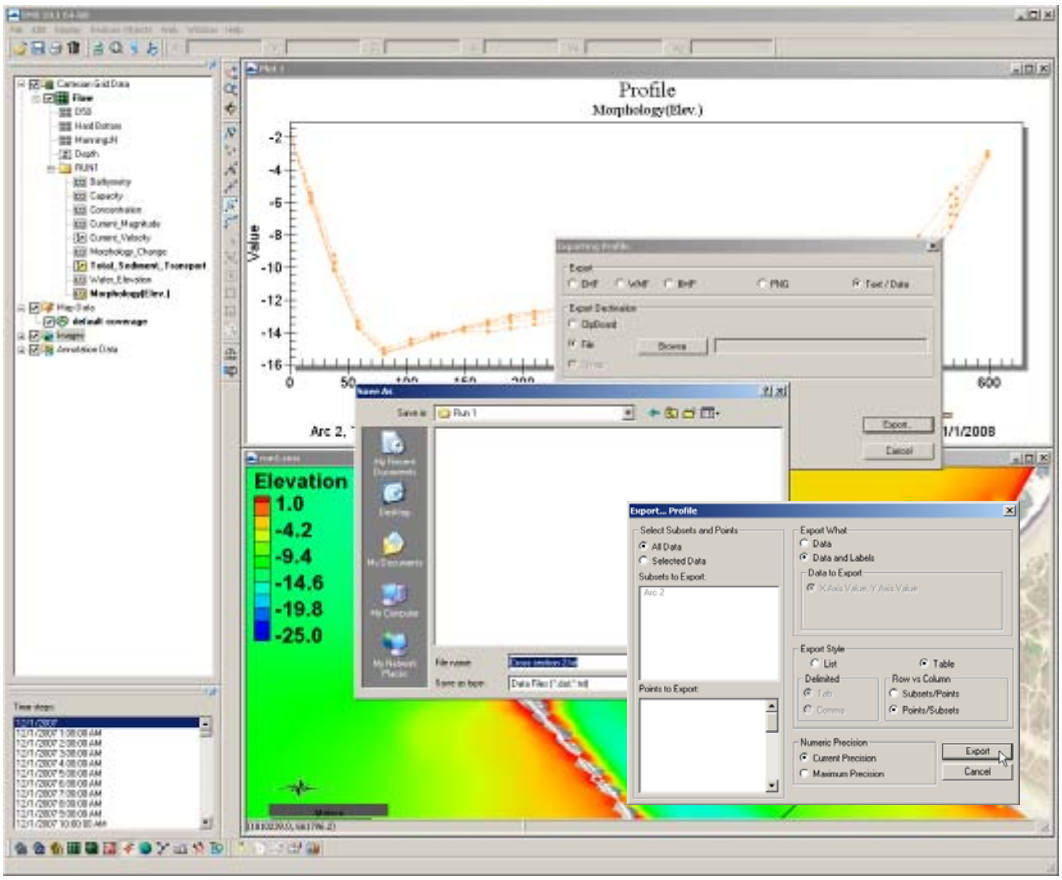


Figure 67. ***Data Export*** **options.**

14.e. Calculating Sediment Transport Rates and Budgets with the CMS

1. Introduction

Calculating accurate sediment transport rates is necessary for predicting channeling infilling, inlet and nearshore morphology change, and on a more basic level understanding sediment pathways and formulating sediment budgets. In the CMS, sediment transport statistics are calculated across predefined 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.

2. Input Files

Load the *HB-Workshop__Flow.cmcards* into SMS by dragging it into the SMS or by clicking File | Open | *HB-Workshop__Flow.cmcards.* The solution file should load in automatically.

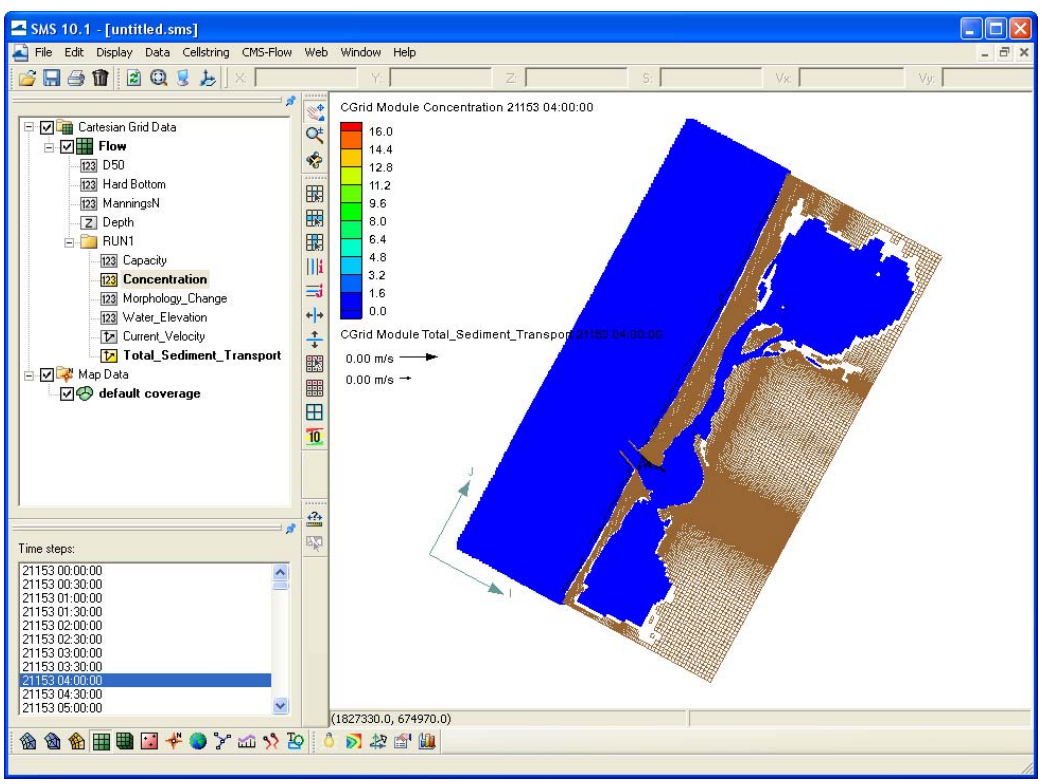


Figure 68. Humboldt Bay CMS-Flow grid with solution file loaded.

3. Model Parameters

1. Create an *Observation Map Coverage*
In the project explorer to the left of the SMS window, either change the type of the *Map coverage* to *Observation* or create a new *Map coverage* of type *Observation*.
a. To change the type of the *Map coverage*
Right-click on the coverage and select *Type | Generic | Observation*

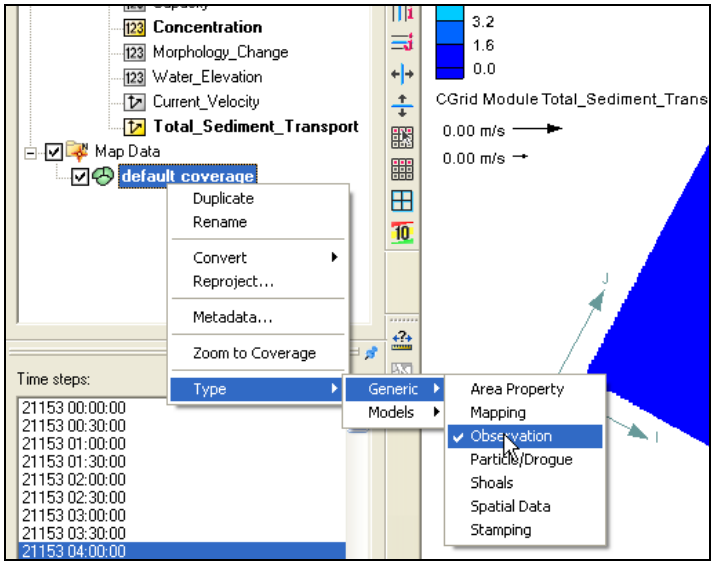


Figure 69. Changing a *Map coverage* type to *Observation.*

b. To create a new Map coverage, right-click on *Map Data* in the project explorer, then select *New Coverage*

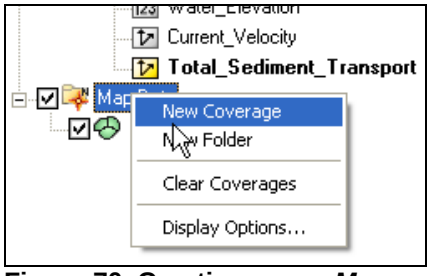


Figure 70. Creating a new *Map coverage* of type *Observation.*

2. Change the name of the *Observation Map Coverage* to Humbolt_Obs_Arcs.

3. Create feature arcs and polygons.
Make sure you are in the Map module by clicking on the *Map Module Tool*

Figure 71. *Map Module* icon in the *Map Module* toolbar in the SMS.

The *Map Module* toolbar should become available in the SMS window. Click on the *Create Feature Arc* tool.

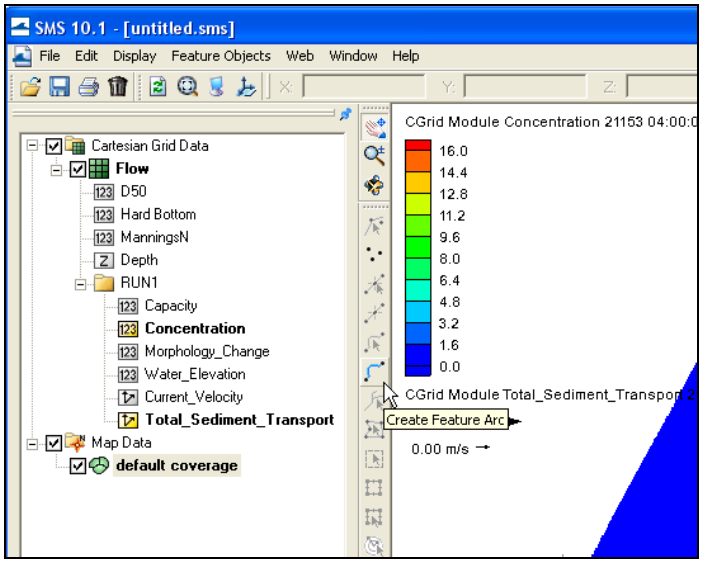


Figure 72. Selecting the *Create Feature Arc* tool.

Click on the starting and ending points on the grid that define the transect for calculating the sediment transport statistics. An example of a *Feature Arc* for calculating the longshore sediment transport at Humboldt Bay is shown in Figure 73. In the case of longshore transport, the feature should extend beyond the breaker zone and closure depth to capture all, mainly, wave-induced longshore sediment transport. Make sure the arc does not touch inactive land cells as this may cause interpolation problems in the SMS.

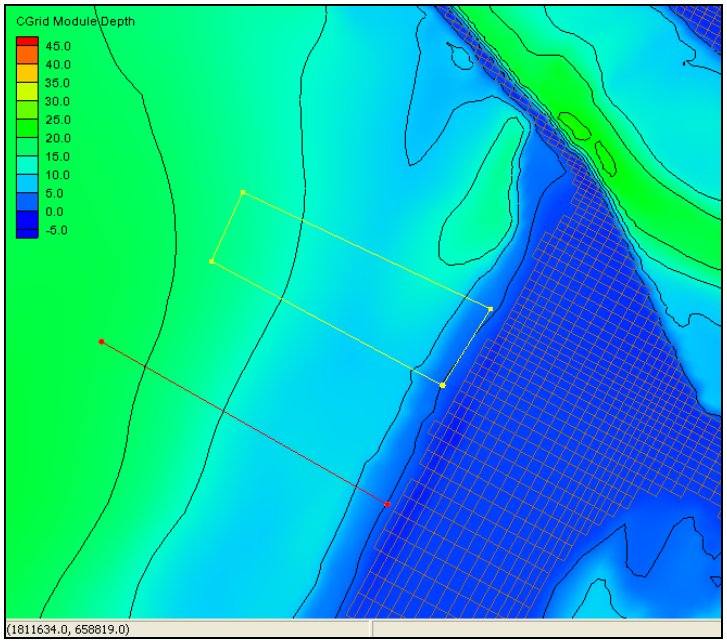


Figure 73. Example of a *Feature Arc* for calculating longshore sediment transport rate.

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 (Figure 74) and right-click on the *Feature Arc* once.

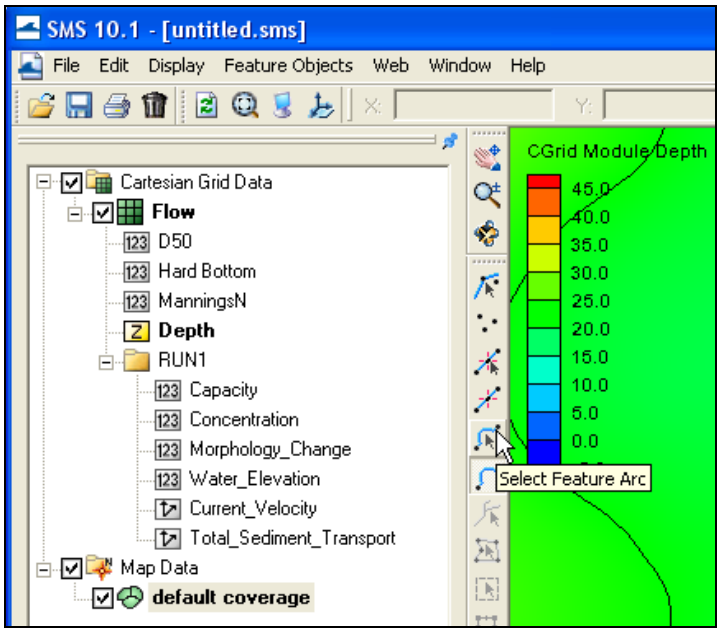


Figure 74. Location of the *Select Feature Arc* tool.

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.

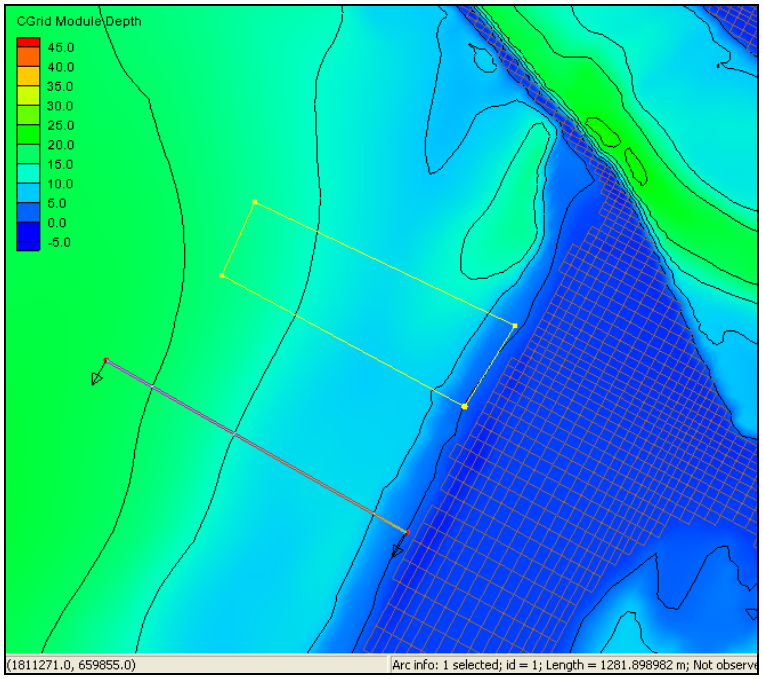


Figure 75. Highlighted *Feature Arc* with the positive direction shown by the arrows.

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 same applies for feature polygons.

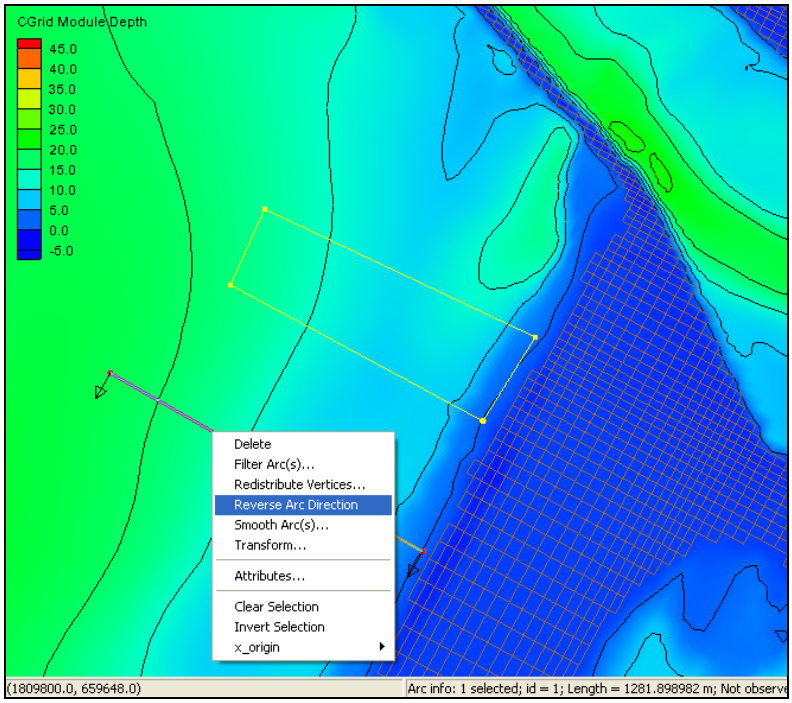


Figure 76. Changing the direction of a *Feature Arc*.

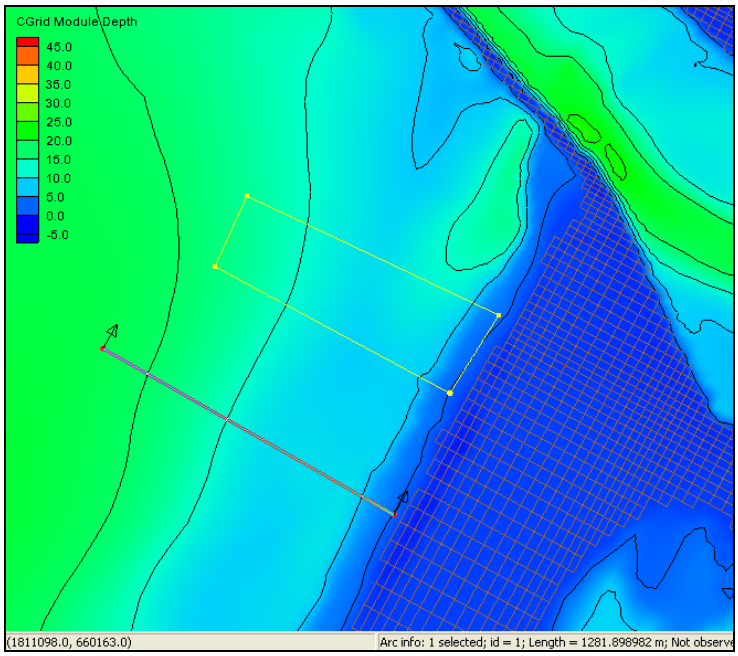


Figure 77. Selected *Feature Arc* with reversed direction.

4. Creating "Ones" Dataset

The sediment transport calculation across a feature arc requires a dataset of the same size and length and constant value of one. To create this dataset, select the Cartesian Grid Module

Figure 78. Location of *Cartesian Grid Module* icon.

Then click on *Data* | *Data Calculator*.

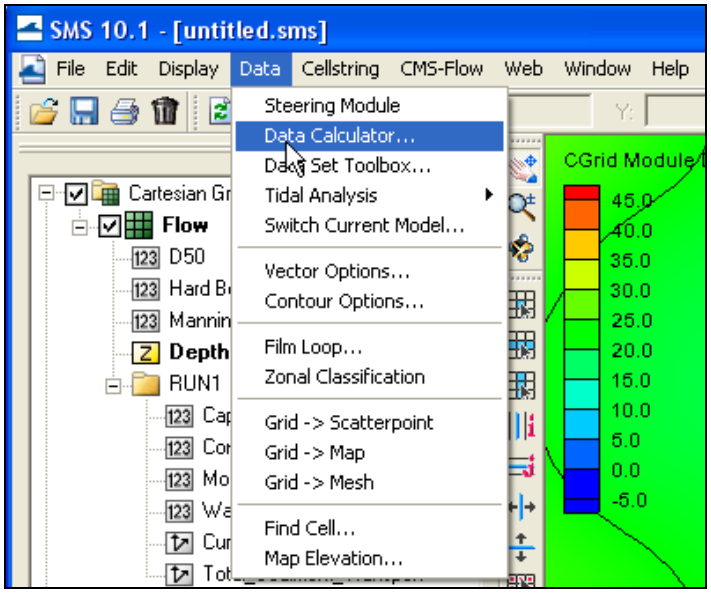


Figure 79. Opening the *Data Calculator* in the SMS.

Once the Data Calculator window opens, follow the steps outlined below:

1. Select the *water elevation* dataset.
2. Click on the *Use all time steps* checkbox.
3. Change the name of the *output dataset* to *ones*.
4. Double-click on the *water elevation* dataset so that it appears in the *Calculator*.
5. In the *Calculator*, multiply the dataset by zero and then add one.
6. Click on *Compute*. The calculation might take a few minutes if the dataset is large. Once the calculation is completed, the output dataset will appear under the *Datasets* window.
7. Click on *Done* to exit the window.

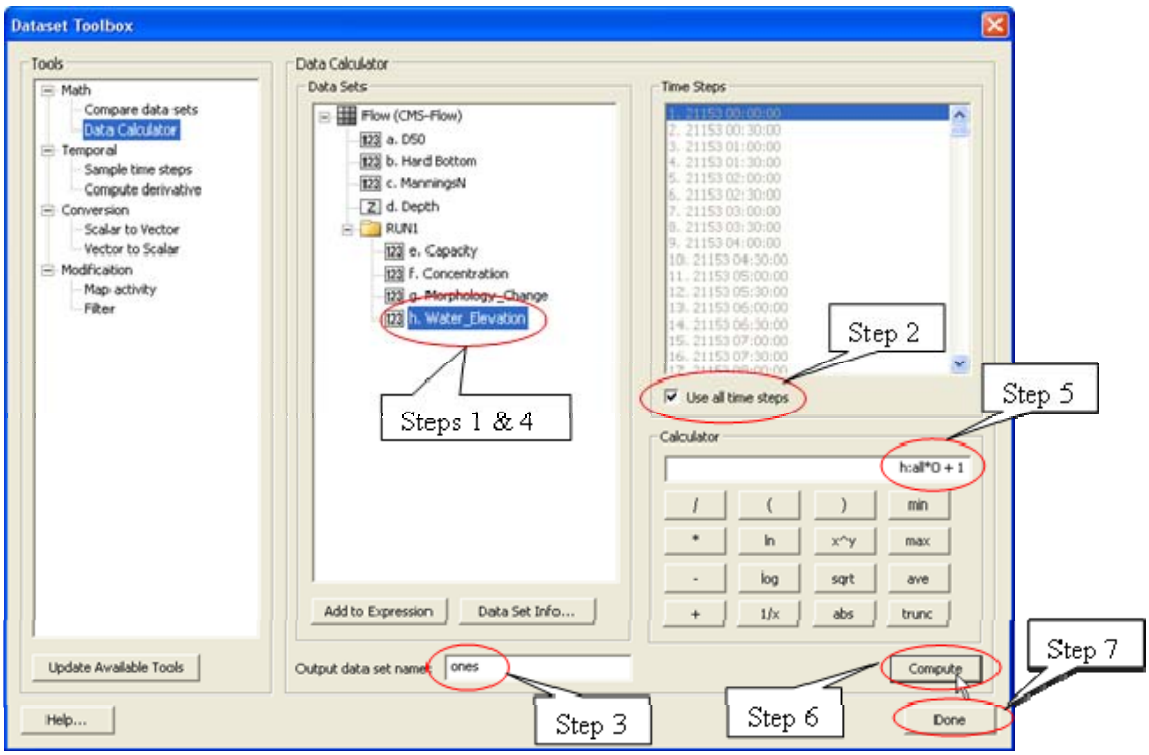


Figure 80. *Data Calculator* window with steps for creating ones dataset highlighted.

5. Sediment Transport Integration

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 in 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:

1. Open the *Plot Wizard* by clicking on *Display* | *Plot Wizard*

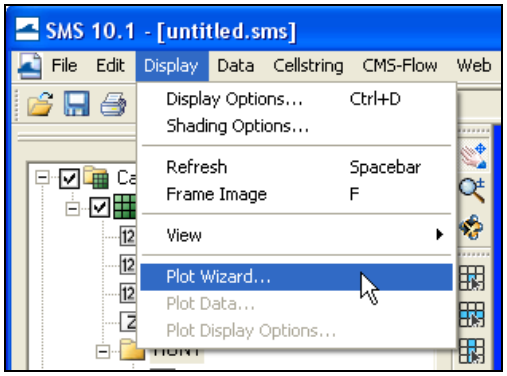


Figure 81. Opening the *Plot Wizard* from the *Display* Menu.

or clicking on the *Plot Wizard icon*

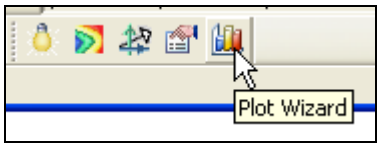


Figure 82. Opening the *Plot Wizard* with *Plot Wizard* toolbar.

2. In the *Plot Wizard – Step 1 of 2* window, select *Time series* as the Plot Type and then click *Next>*.

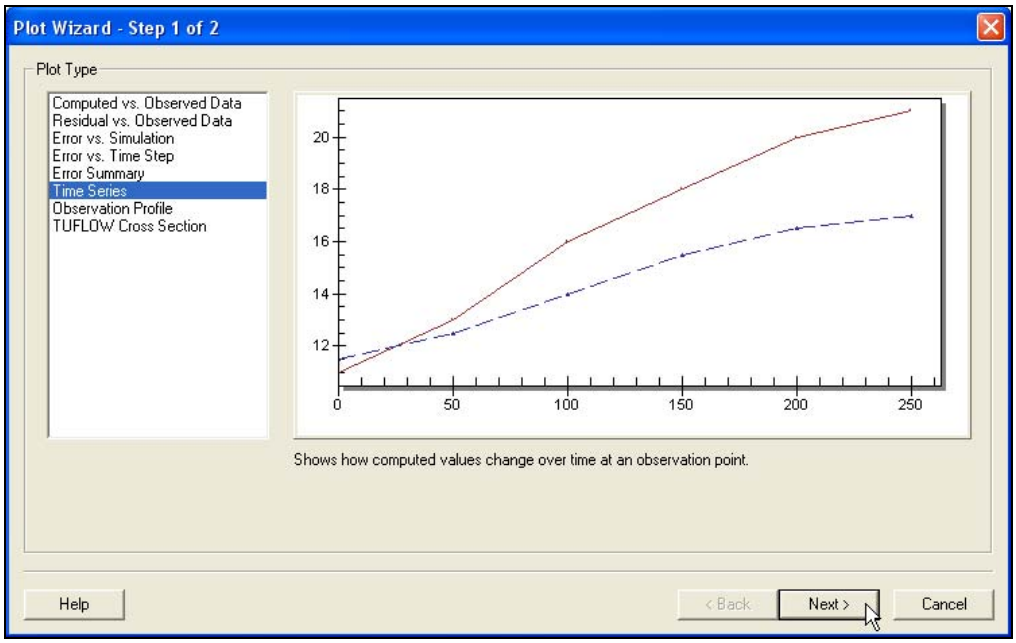


Figure 83. Step 1 of 2 of the *Plot Wizard.*

3. In the *Plot Wizard – Step 2 of 2* window, select *Flux* as the *function type*.

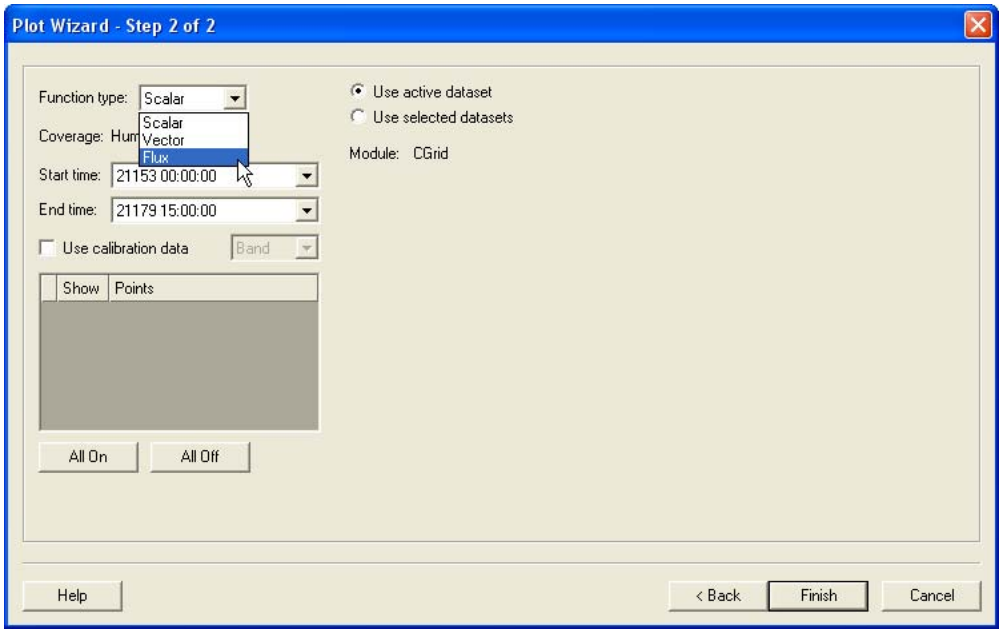


Figure 84. Selecting the *Function Type* as *Flux* in Step 2 of 2 of the *Plot Wizard.*

4. Click on the feature arc checkbox. Select the *ones* dataset for the scalar dataset, and the sediment transport as the vector dataset. Make sure the start and end times are correct. Click *Finish.*

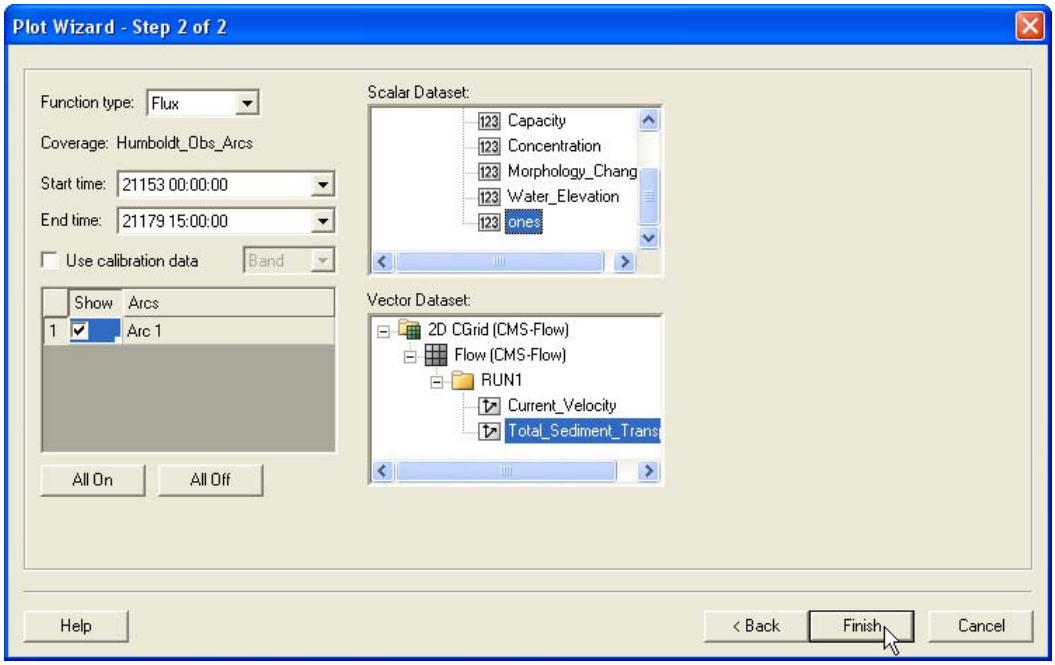


Figure 85. Selecting the correct *Start* and *End Times*, *Feature Arcs*, and *Scalar* and *Vector Datasets* in Step 2 of 2 of the *Plot Wizard.*

The computation might take a few minutes, especially if the dataset is large.

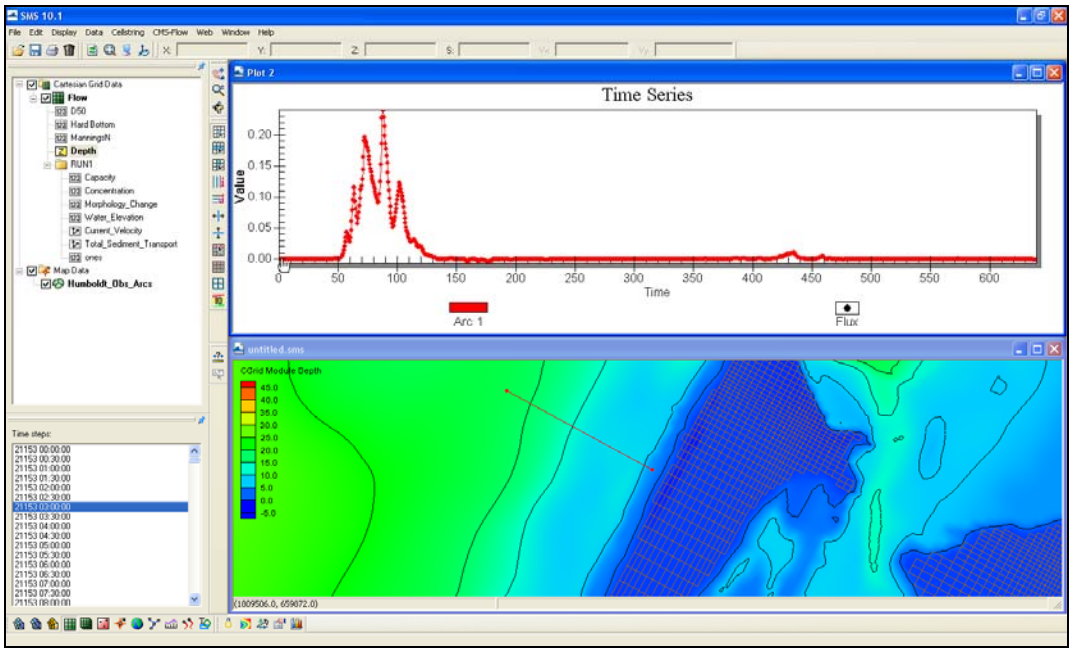


Figure 86. Example time series of integrated sediment transport across a *Feature Arc.*

5. Right-click on the time series plot and select *View values…*

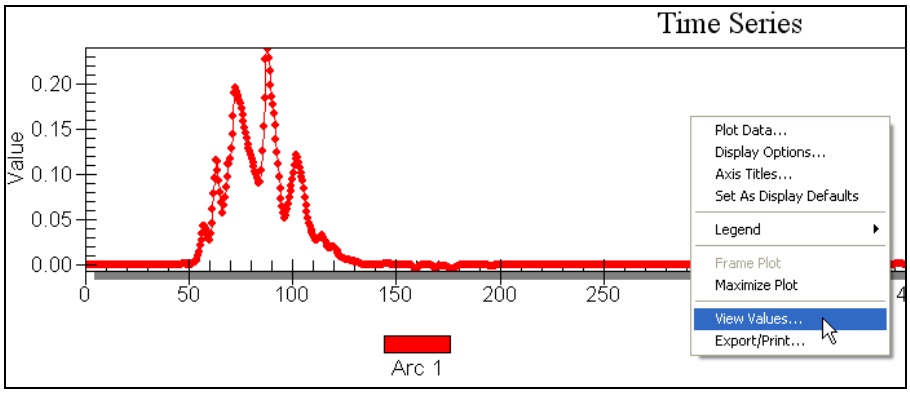
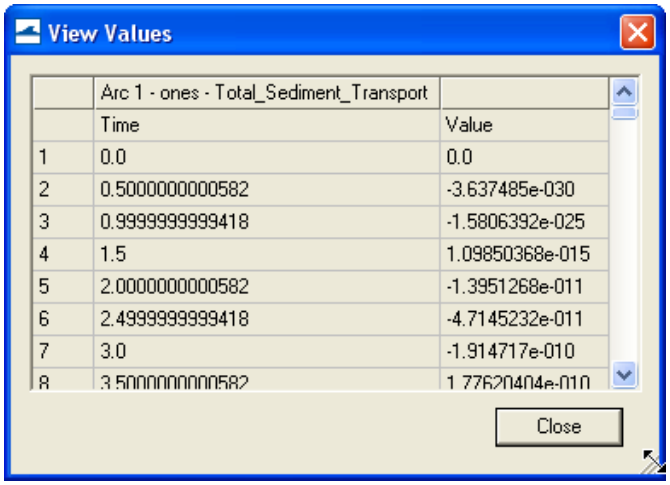


Figure 87. Selecting the View Values option after right-clicking on the time series window.

A table will be displayed with the time in hours and integrated sediment transport rate across the feature arc.



	Arc 1 - ones - Total_Sediment_Transport	
	Time	Value
1	0.0	0.0
2	0.5000000000582	-3.637485e-030
3	0.9999999999418	-1.5806392e-025
4	1.5	1.09850368e-015
5	2.0000000000582	-1.3951268e-011
6	2.4999999999418	-4.7145232e-011
7	3.0	-1.914717e-010
8	3.5000000000582	1.77620404e-010

Figure 88. Example of a *View Values* window in SMS.

6. Calculating Sediment Transport Statistics

There are two ways of calculating the sediment transport statistics. One is with a Fortran executable, and the other is with an Excel spreadsheet.

1. From the *View Values* window (see Section 5), select the data from the data 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. Create a header for the file as shown below where the second number is the number of rows in the data, and the text in between quotes is a user-specified name. Save the file with the extension *xys*.

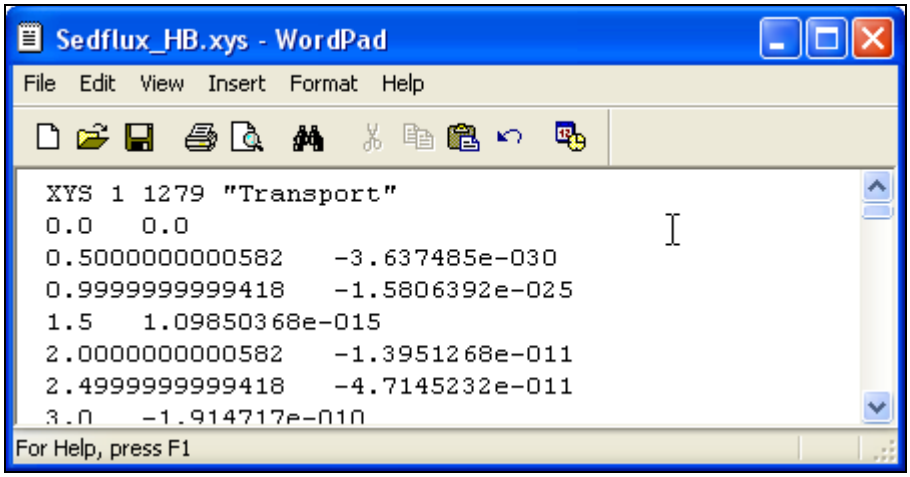


Figure 89. Example of XYS file with extracted sediment transport time series.

2. Make sure you have a copy of proc_flux.exe in the same directory as the xys file.

3. Open a command prompt by going to *Windows Start* button and then selecting *Accessories | Command Prompt.*

4. Change directories to the folder which contains the xys file and the proc_flux.exe program.

5. Type in at the command prompt:

```
> proc_flux.exe  input.xys  output.txt
```

6. Press enter when the executable is finished.

An example of the command prompt showing the steps for calculating the sediment transport statistics with the *proc_flux.exe* application is shown in Figure 90.

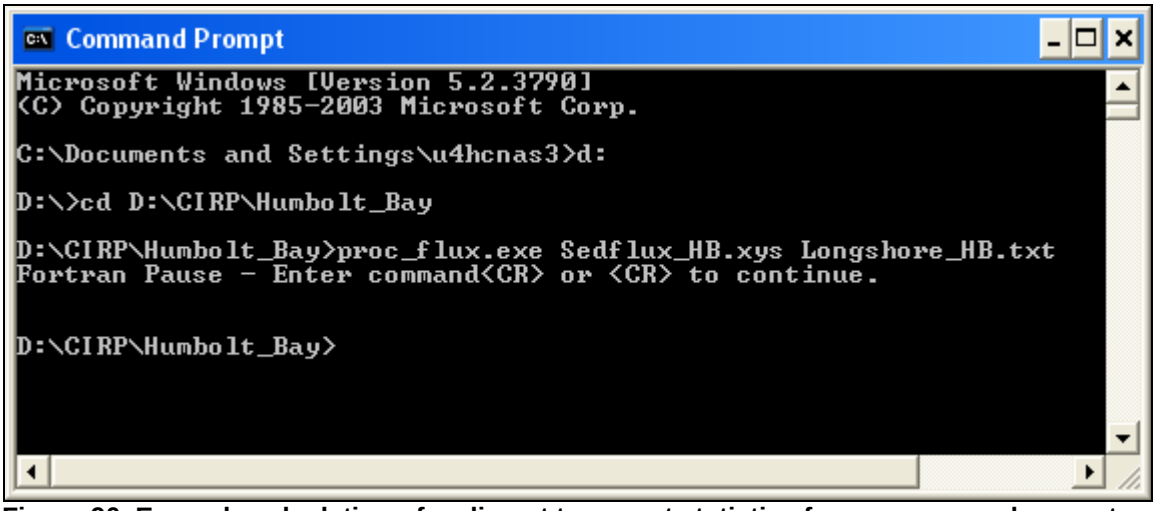
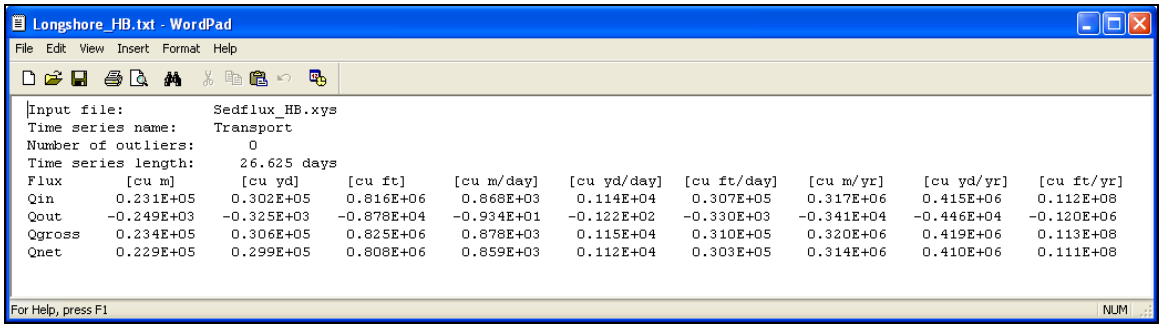


Figure 90. Example calculation of sediment transport statistics from a command prompt.

If no output file is specified, the default output file name is out.txt. An example of the output file is shown in Figure 91.




```
Input file:          Sedflux_HB.xys
Time series name:    Transport
Number of outliers:      0
Time series length:     26.625 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       0.231E+05    0.302E+05    0.816E+06    0.868E+03    0.114E+04    0.307E+05    0.317E+06    0.415E+06    0.112E+08
Qout     -0.249E+03   -0.325E+03   -0.878E+04   -0.934E+01   -0.122E+02   -0.330E+03   -0.341E+04   -0.446E+04   -0.120E+06
Qgross    0.234E+05    0.306E+05    0.825E+06    0.878E+03    0.115E+04    0.310E+05    0.320E+06    0.419E+06    0.113E+08
Qnet      0.229E+05    0.299E+05    0.808E+06    0.859E+03    0.112E+04    0.303E+05    0.314E+06    0.410E+06    0.111E+08
```

Figure 91. Example output file of *proc_flux.exe* with sediment transport statistics.

14f. Breaching and Morphology Change near Jetties[2]

This hands-on CMS practice covers the breaching of an idealized barrier island under combined storm surge and waves. Breaching at jetties can be simulated in a similar manner. The breaching is calculated through the steering operation (coupling CMS-Wave and CMS-Flow) using the SMS, because overwash and breaching can occur by wave run-up and by inundation. A pre-generated CMS-Flow Grid, island-breaching.cmcards, is used in this practice. The steering is conducted for a 10-hr storm surge simulation with spectral wave input at 2-hr interval. The CMS-Flow grid consists of 110 x 101 square cells, with each cell of 20 m x 20 m. A constant grain size of 0.2 mm and constant Manning's n (friction coefficient) of 0.025 are specified for the simulation.

The initial barrier island divides the CMS-Flow domain into two separate water bodies. The seaward wave input boundary has a constant depth of 14.5 m and the bayside water has a constant depth of 3 m for this exercise. The barrier island has a short middle section with lower crest elevation of 3 m and two side long sections with higher crest elevation of 4 m. The seaside boundary water surface escalates from 0 to 3 m in the first 4 hours, remains at a constant of 3 m in the next 2 hours, and recedes gradually to 0 m (sea level) in the last four hours of simulation (Figure 92). The bayside water level is kept at 0 m in the simulation (assuming the bay is sufficiently large that the bayside water level change is negligible as the sea water flows into the bay during the breaching condition) during this short storm. The CMS could also calculate the water level in the bay simultaneously with the surge if it were open to the ocean as through a nearby inlet.

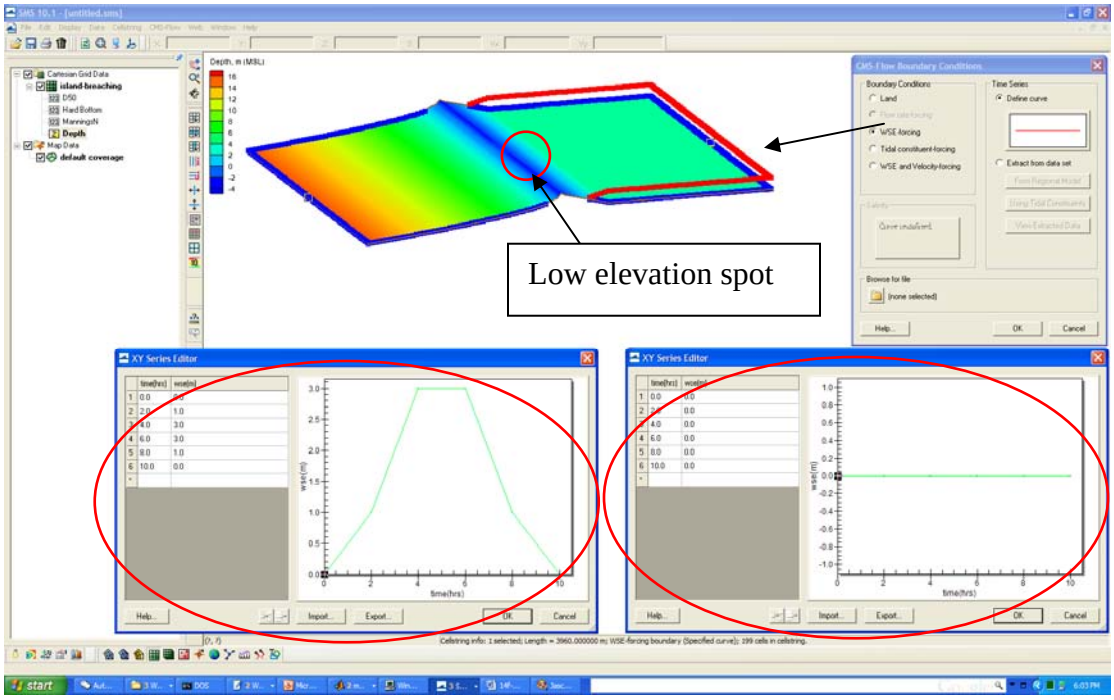


Figure 92. CMS-Grid with the water level input along the seaside and bayside boundaries.

[2] The breaching sample files are located in the "Hands-on/Breaching and Morphology Change" directory.

The CMS-Wave grid is transformed from the CMS-Flow grid.

10. Open the CMS-Flow file, island-breaching.cmcards,

11. Choose *Data | Switch Current Model*,

12. Select *CMS-Wave* and click *OK*,

13. Click *Select grid cell* tool and select cells with the depth < 2 m (use *Edit|Select by Data Set Value*),

14. Right click the highlighted area of the select cells and choose the *Cell Attributes*,

15. Select Wave run-up in the *Cell Attributes|Type* and *OK*,

16. Choose *CMS-Wave | Spectral Energy*,

17. Click *Create Grid* and *OK*,

18. Click *Generate Spectra* and choose *TMA (shallow water) spectrum*,

19. Specify 14.5 m for *Gauge Depth* and choose *Shore Normal* wave for the input wave direction *Projection*,

20. Type the wave label (1,2,3,.., 7), wave angle (0 deg), wave height (1, 1.5, 2.5, 2.5, 1.5, 1, 1 m), wave period (6, 6, 8, 8, 6, 6, 6 sec), gamma (3.3), and directional spreading (100),

21. Click *Generate* and *Done*,

22. Use *File|Save as* to save the CMS-Wave grid, island-breaching.sim,

23. Open (load) island-breaching.cmcards,

24. Use *Data|Steering Module* to start the *Steering Wizard*, and click *Next*,

25. Type 2 hours for wave input interval (*Run CMS-Wave every*:) and check *Current Field*, *Water Surface Elevation*, and *Water data*,

26. Type 200 m for *Extrapolate out* and *Start* the steering operation.

Figure 93 to Figure 97 give the pictorial summary of the above steps. Figure 98 shows the breaching morphology change after the 10-hr simulation. (Note: without wave run-up, the steering would not trigger breaching of the barrier island for this case.)

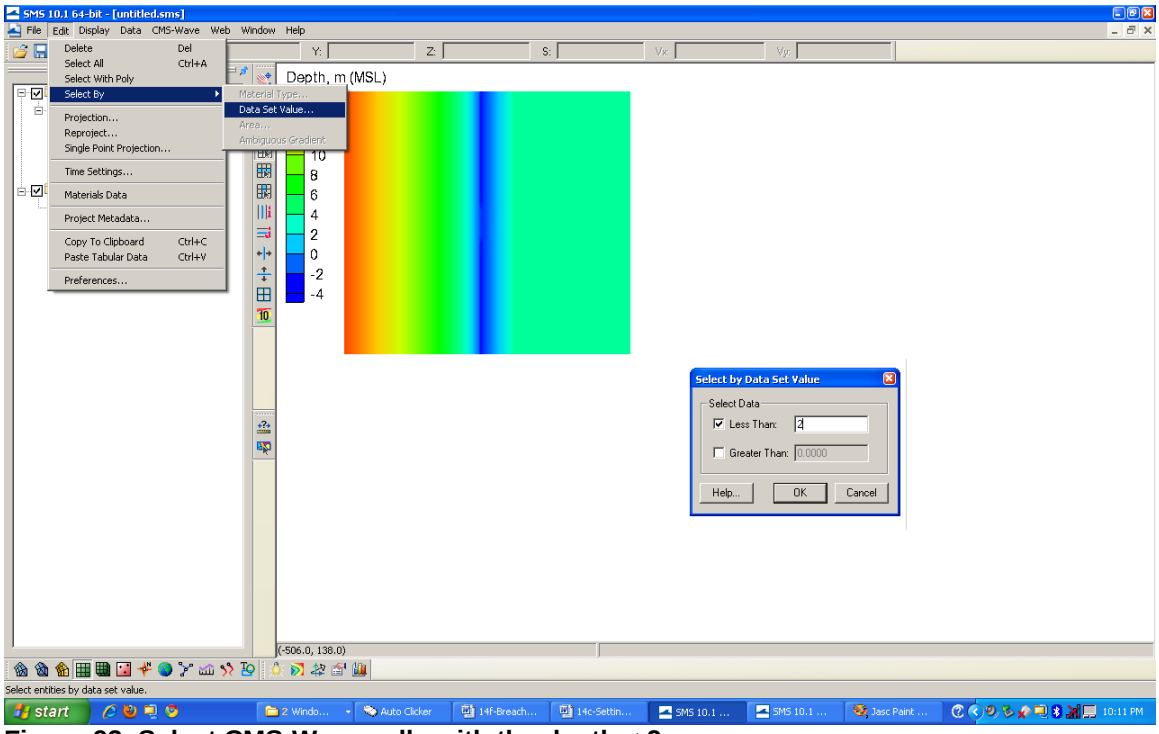


Figure 93. Select CMS-Wave cells with the depth < 2 m.

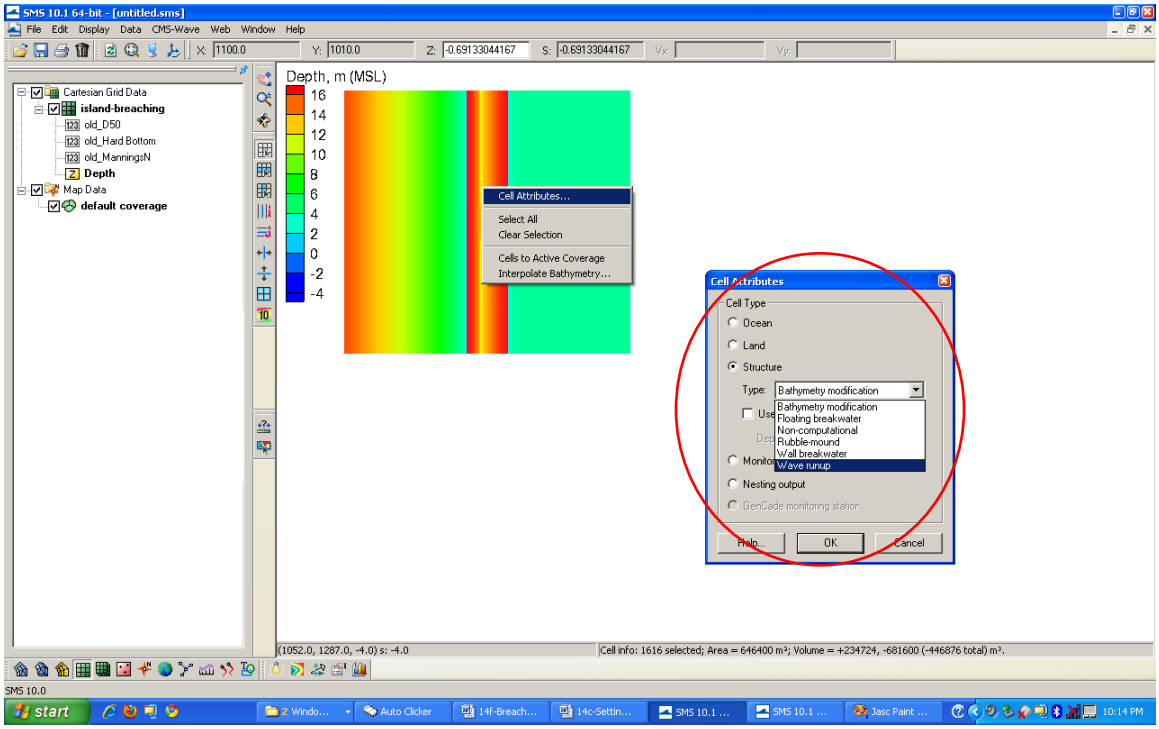


Figure 94. Select Wave run-up in the *Cell Attributes|Type*.

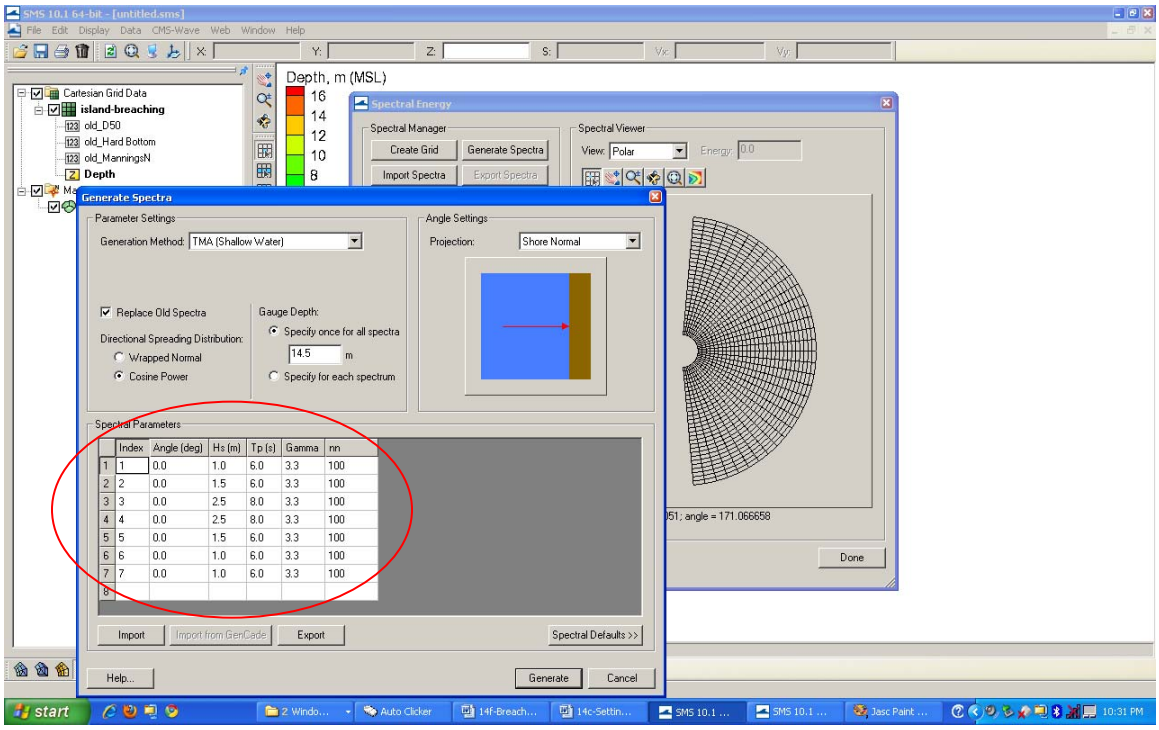


Figure 95. Choose *TMA (shallow water) spectrum* and specify the input spectral parameters.

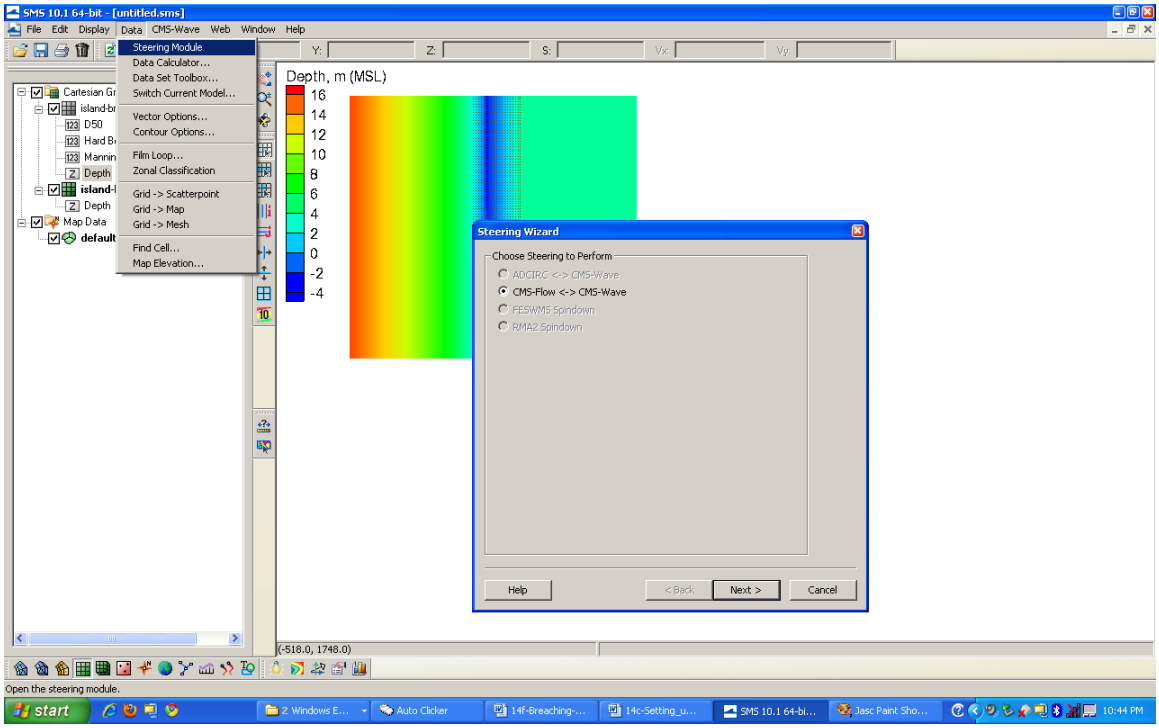


Figure 96. Use *Data|Steering Module* to start the *Steering Wizard*.

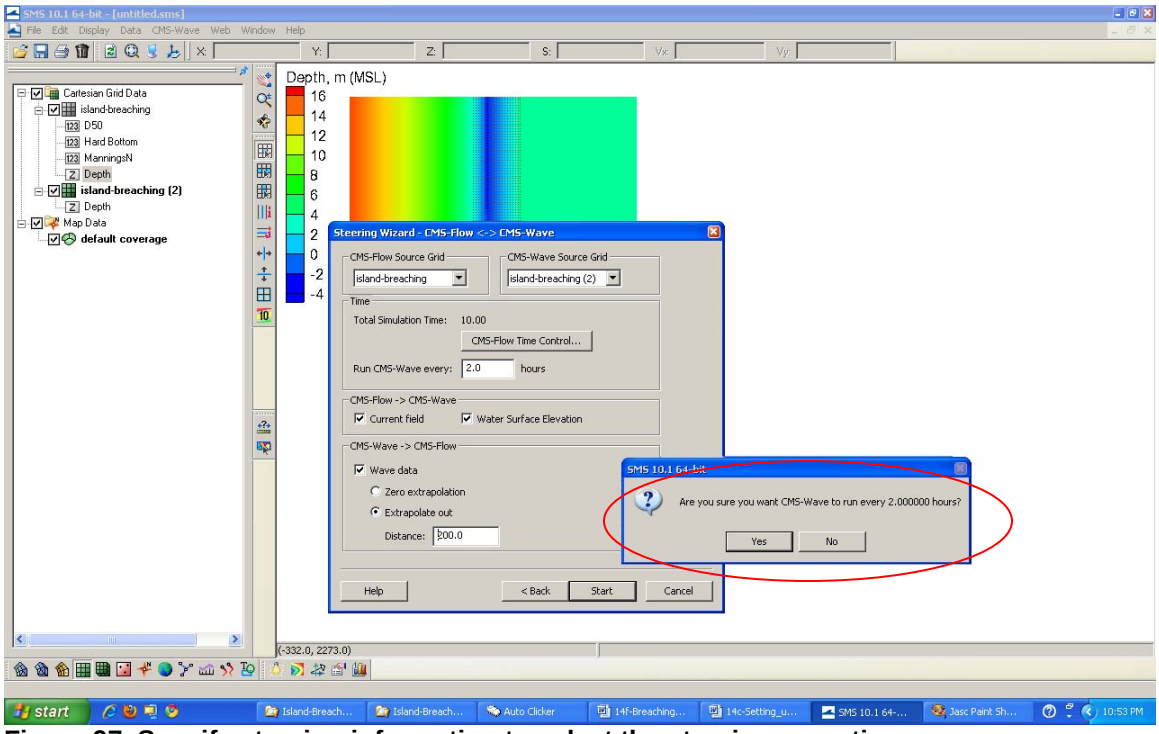


Figure 97. Specify steering information to select the steering operation.

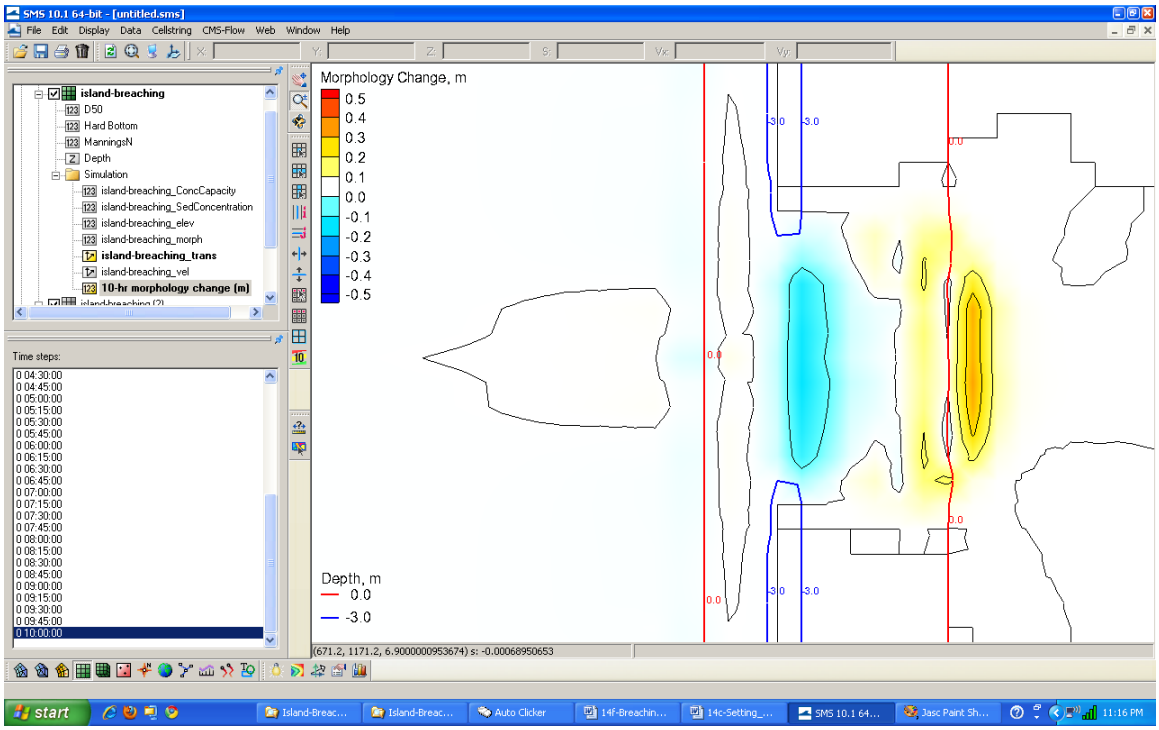


Figure 98. Calculated barrier-island breaching morphology change after 10-hr of combined storm surge and waves.