Instruction with Hands-on Practice:
Grid Generation and Forcing

Introduction

For this section of the hands-on tutorial, we will be using a merged dataset of bathymetry points.   You will need the following files, located in the "CMS-Humboldt" directory:

- HB-Grid – Merged.h5
- HB-Grid.cmcards
- HB-Grid_grid.h5
- HB-Grid_mp.h5
- Humboldt_WSE.xys
- Humboldt_flow.xys

Set Model Projection

The merged dataset that will be used for this hands-on demonstration was created in a different zone than the one that you have previously been working with. We must now change this projection to match our new location.

To accomplish this, click on *Edit*, *Projection* (Figure 1), and set the *Global Projection* to State Plane Coordinate System, NAD83 *Datum*, Meters *Units*, California Zone 1(FIPS 401) *Zone*, and the *Vertical Projection Units* to Meters, then click *OK*.
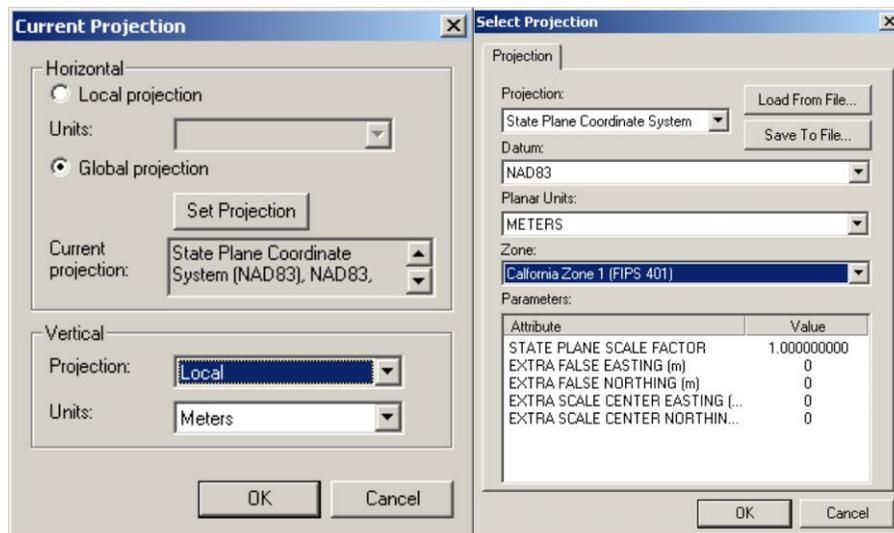

Figure 1. Projection dialogs.

Remove Long Triangles

Load the merged bathymetry dataset by selecting *File | Open* from the pull-down menu. Then browse to the "CMS-Humboldt" directory and click the file "HB-Grid Merged.h5".

Long triangles are an indication of missing data points. Many times the grid that is generated after removing long triangles will look better. For our grid domain, we can remove triangles that have an edge that is longer than 2000 meters. To do this, click *Scatter | Scatter Options* (Figure 2), and change the "Max. edge length" to 2000.
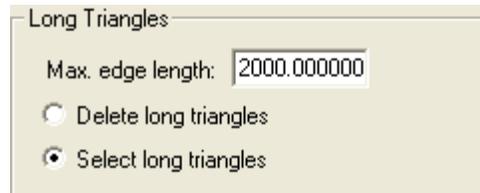

Figure 2. Long triangle dialog box.

Next, click *Triangles, Select Long Triangles*, wait for the SMS to finish selecting them, and then click delete.

Generating a Flow Grid

The CMS presently has two grid options it can use for calculations: either constant, regularly spaced cells; or refined, rectangular, irregular cells. The refined option is valuable for optimizing (minimizing) CMS run time. However, hydrodynamic and sediment transport time steps are dependent upon cell size in comparison to water depth and, therefore, the user has the challenge of minimizing the number of grid cells while providing good resolution in the vicinity of channels, structures, morphology, and other features of interest.

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 3), and set the base cell size for the area of the grid where the points are located.
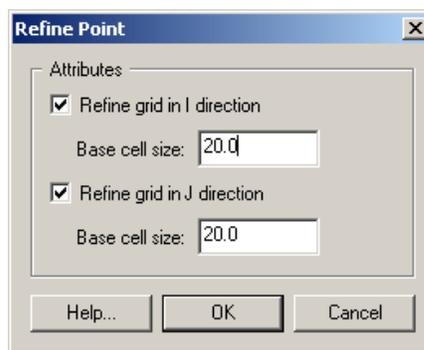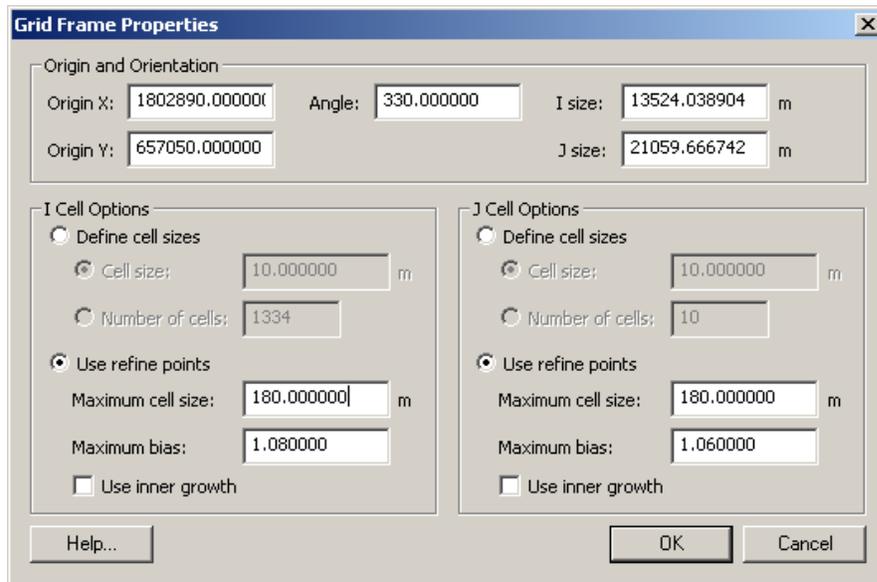

Figure 3. Refine point options.

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 4),

Figure 4. 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 5) 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 of the I and J directions do not need to be specified for each refine point, if refining only one direction is required.
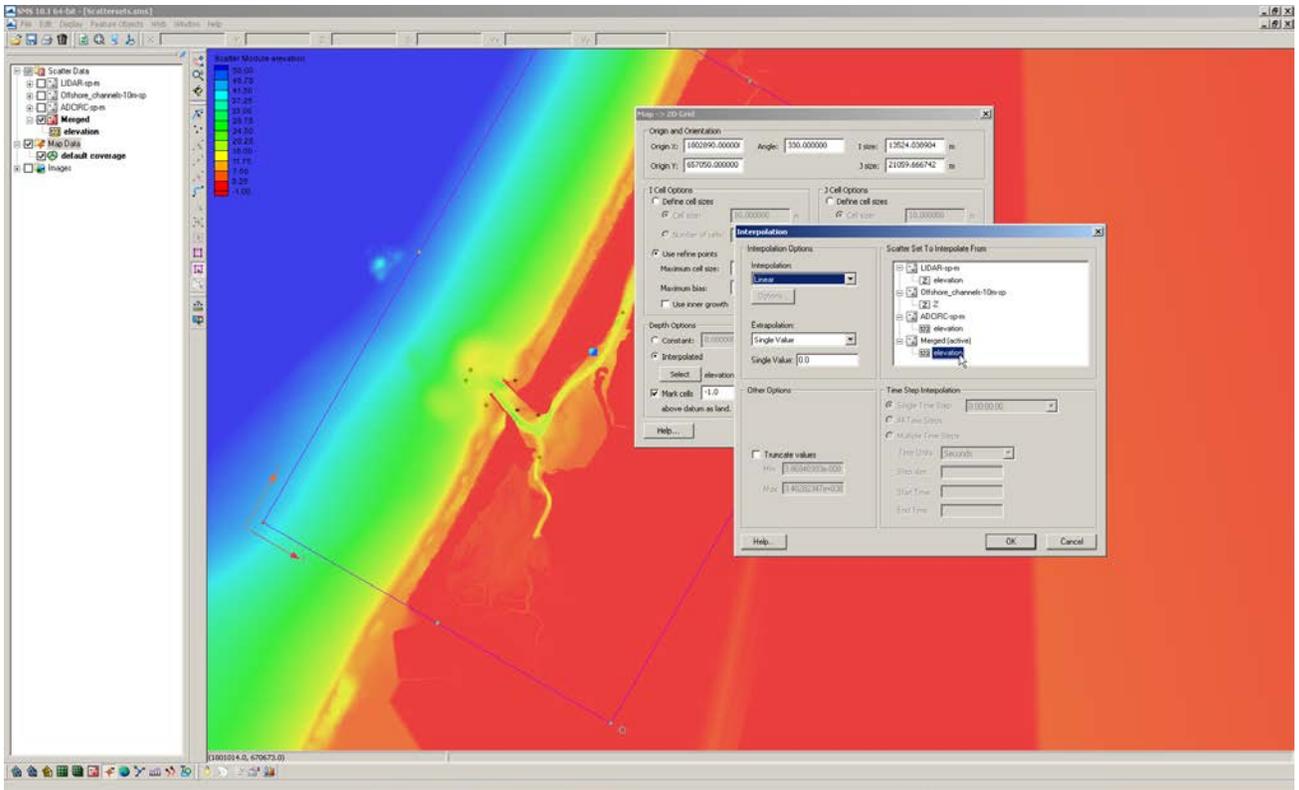
**Figure 5. Depth interpolation options for the Map -> 2D function.**

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 should not 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 6), or manually select cells by dragging the mouse over the target ocean cells and change the Z,

Figure 6. 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 7), and change the cell type from *Active Ocean* to *Inactive Land* cells, *OK*,
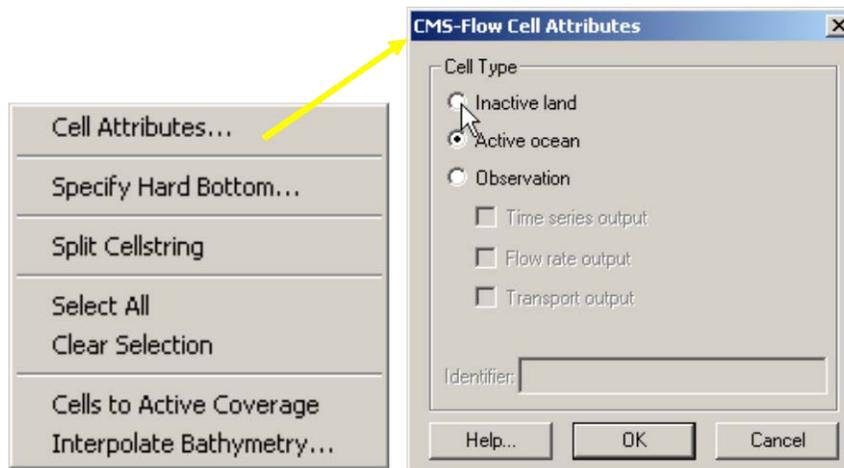
Figure 7. CMS-Flow Cell Attributes Dialog

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.

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 "CMS-Humboldt" directory and click the file "HB-Grid.cmcards" (shown in Figure 10).
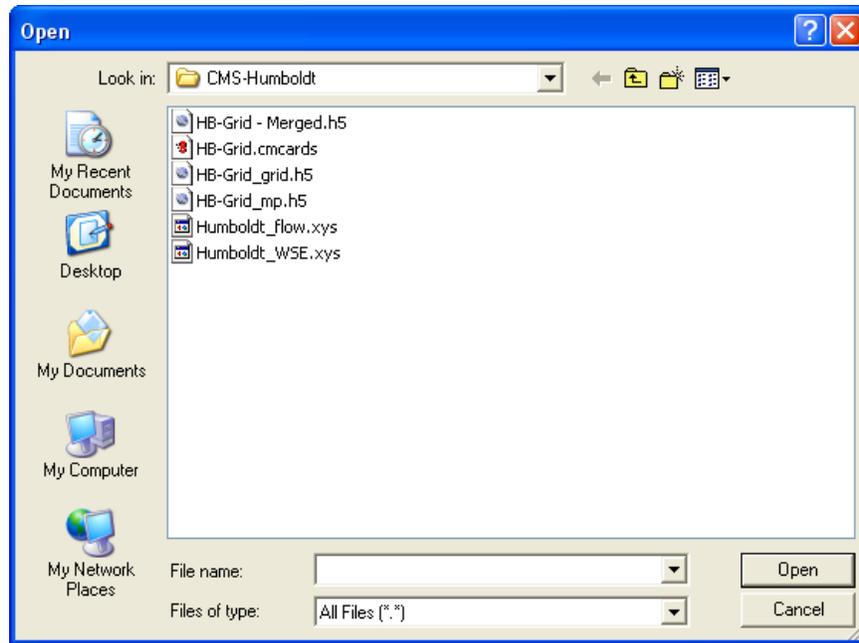
Figure 8. "File | Open" Dialog in SMS.

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. The manual creation of a cellstring for a river flow boundary will be covered later in this section.

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 11. 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.
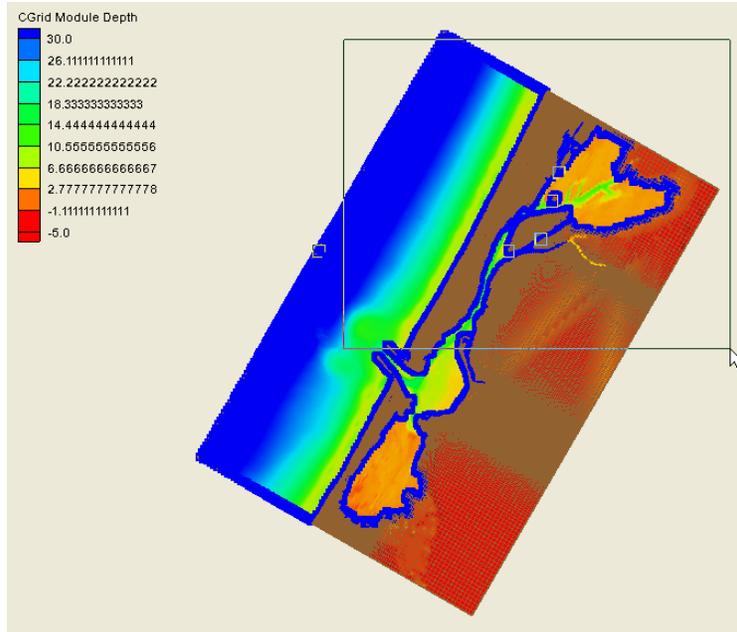
Figure 9. Selection of Land Cellstrings.

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 12 below.
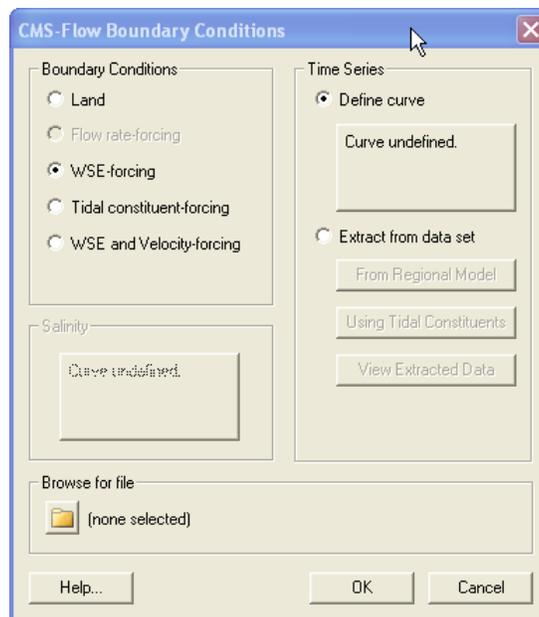

Figure 10. Boundary Condition dialog.

For this option, the choice made in Figure 12, 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 13 below, where a curve can be entered manually, cut and pasted from a spreadsheet, or imported from a file.
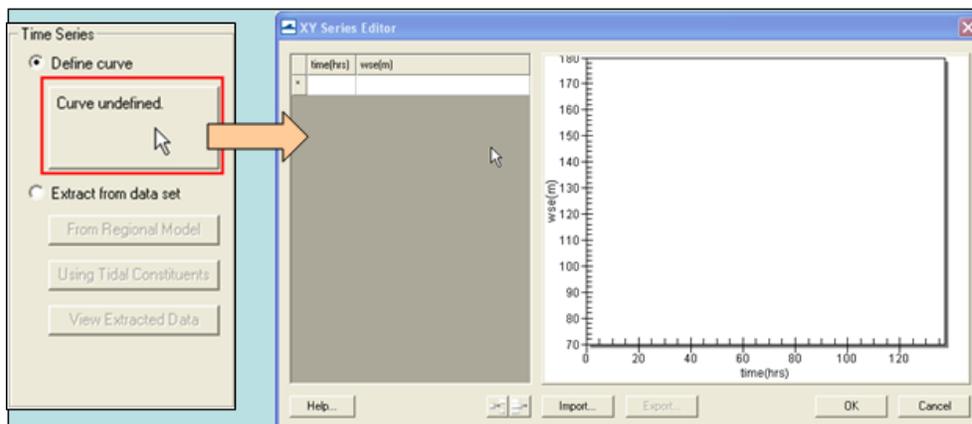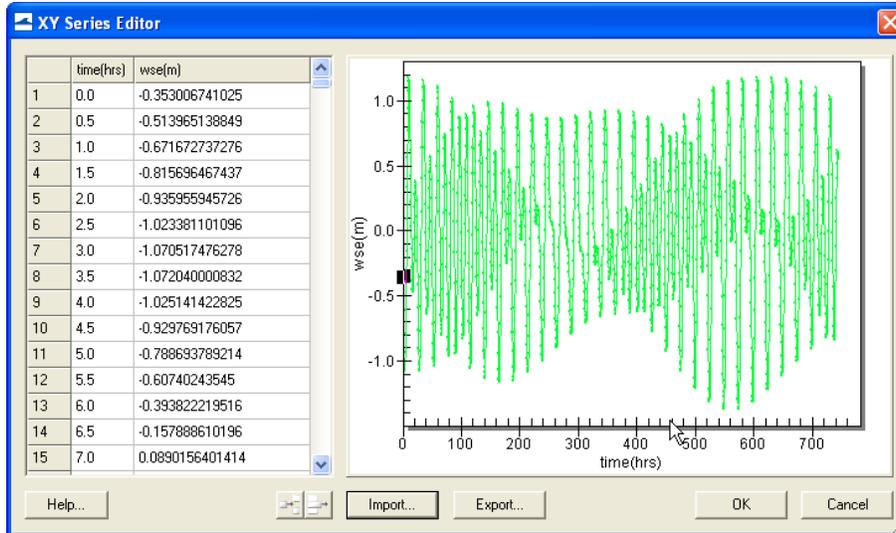

Figure 11. Time series selection.

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 14), click *OK* to close the time series editor and then click *OK* again, to close the Boundary Condition Dialog.

Figure 12. 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 15. 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 16.

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.
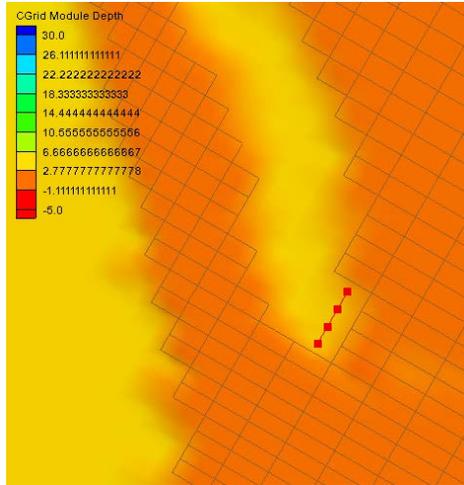
Figure 13. Create river flow cellstring.
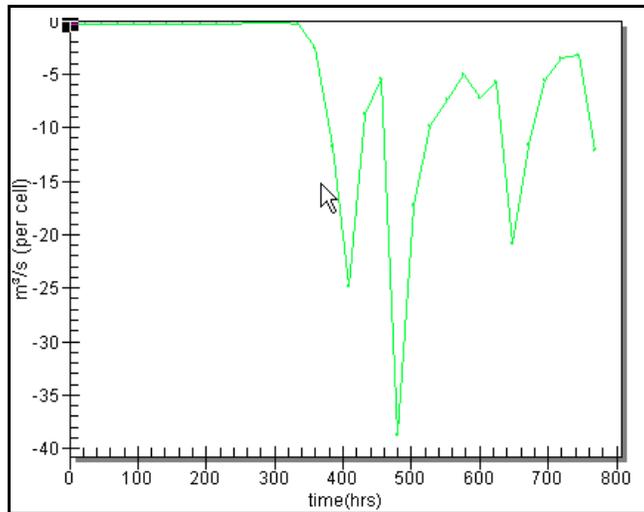
Figure 14. Flow rate curve.

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 17), right-clicking brings up a selection menu. Choose the option "Specify Hard Bottom" and the menu shown in Figure 18 will be displayed.
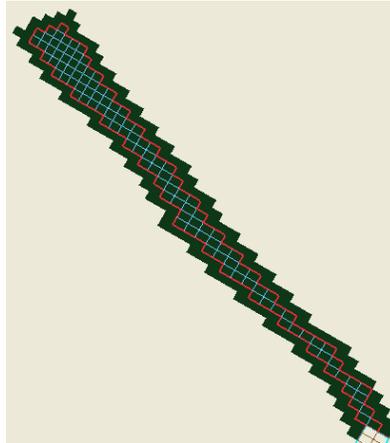

Figure 15. Selected cells for the Humboldt south jetty.
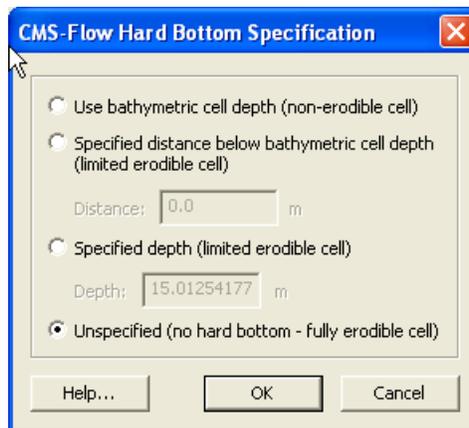

Figure 16. Right-click menu for selected cells.

There are several options on this menu:
1. <u>Use Bathymetric cell depth.</u> This option assigns the Z value (depth) to be the maximum erodible depth. This is the most common choice.
2. <u>Specified distance below bathymetric cell depth.</u> Use this option if you have an idea of the amount of sand above the hard bottom area.
3. <u>Specified depth.</u> This option assigns the present assigned depth PLUS the entered value as the maximum erodible depth for each cell.
4. <u>Unspecified.</u> 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 19.

Figure 17. 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 20) 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 21 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.
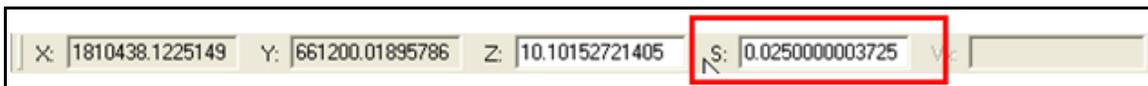
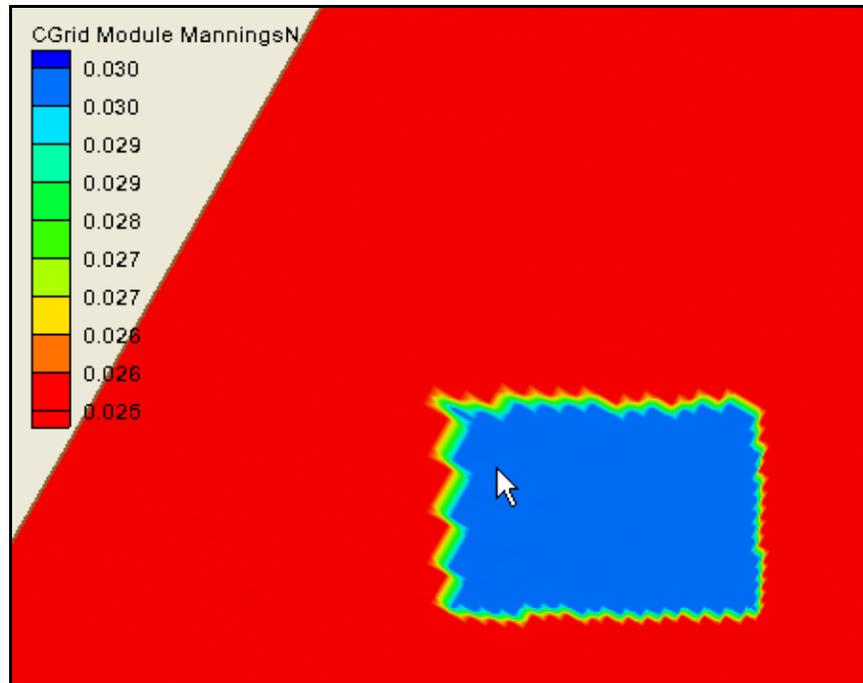**Figure 18. Scalar dataset edit box.**

Figure 19. 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 22) consists of various pages (or tabs) of related information. This section will cover the following tabs:
1. Model Parameters
2. Wind/Wave
3. Output
4. Advanced
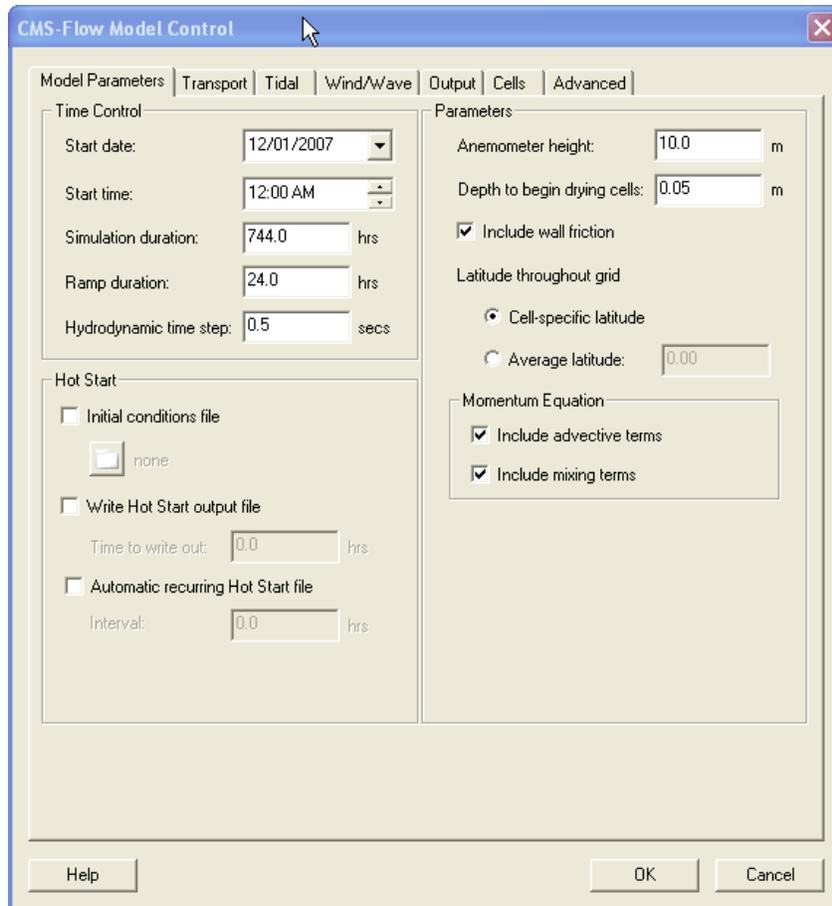
Figure 20. CMS-Flow Model Control.

Model Parameters Tab

1. <u>Start Date</u> – Beginning date of simulation. Used in CMS-Flow to define the starting date of the simulation.
2. <u>Start Time</u> – Generally, simulations should start at 0000 hr GMT, but could be changed to any time of the day.
3. <u>Simulation Duration</u> – 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.).
4. <u>Ramp Duration</u> – 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.
5. <u>Hydro time step</u>?
6. <u>Initial Conditions File</u> – 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.
7. <u>Write Hot Start output file</u> – CMS-Flow will write out a single hot start file after the entered number of hours have passed in the simulation.
8. <u>Automatic recurring Hot Start file</u> – CMS-Flow will write out a hot start file at a specified interval (i.e. Every two hours)

9. <u>Anemometer height</u> – Height that the anemometer was installed, if wind forcing has been selected.
10. <u>Depth to being drying cells</u> – 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.
11. <u>Include wall friction</u> – Additional friction is included for cells next to land.
12. <u>Latitude throughout grid</u> – Two options are available:
    a. Cell-specific latitude – each cell has its own latitude value (default)
    b. Average latitude – CMS-Flow assumes there is no variability in the latitude for a given grid.
13. <u>Momentum Equation</u>
    a. <u>Include advective terms</u> – Switch to turn on/off advective terms in the model run.
    b. <u>Include mixing terms</u> – Switch to turn on/off mixing for the model run.

Wind/Wave Tab

Wind data:
1. <u>Include wind</u> – Switch to turn on/off wind forcing. Further options for wind are only available if this switch it checked.
2. <u>Import wind from file</u> – Import wind information from one file that contains both velocity and direction information.
3. <u>Velocity (m/s)</u> – Edit box to specify/import wind velocity information.
4. <u>Direction (deg</u>.) – 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:
1. <u>Include radiation stresses</u> – Switch to turn on/off radiation stress forcing from a file mapped to CMS-Flow grid.
2. <u>Include wave conditions –</u> 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

1. <u>Output Times Lists</u> – 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.
2. <u>Add by time step</u> – 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).
3. <u>Output Datasets</u> – 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."
4. <u>Simulation Label</u> – 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.

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. Parameters with non-default values are listed here.
Model Parameters Tab
Start date: 12/01/2007
Simulation duration: 744 hrs
Hydrodynamic Time Step: 0.5 seconds (Explicit) <u>or</u> 600 seconds (Implicit)
Number of Threads: 2 (or larger, depending on your computer)
Output Tab
Output Times 1
Start Time: 0.0 hrs
Increment: 0.5 hrs
End Time: 744 hrs
Output Datasets
Water surface elevation – Output Times 1
Velocity – Output Times 1

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