Particle Tracking Model (PTM) with a Quadtree CMS Grid

Overview

A step by step PTM setup is demonstrated through a CMS application with a telescoping grid at Humboldt Bay, CA. Humboldt Bay is a natural multi-basin, bar-built, coastal lagoon located along the North Coast of California. A pair of rubble-mound jetties protects the entrance from the high-energy waves of the Northeast Pacific coast. Waves, tides, and coastal currents are the primary forcing to drive local sediment transport in the surrounding region (Figure 1).

Figure 1. Humboldt Bay, CA.

Humboldt Bay has a hazardous harbor entrance that requires constant dredging to maintain deep-draft channels. The annual dredging volume, primarily containing fine sand, is approximately 1 million CY. The CMS can be applied with the PTM to investigate dredge-induced sediment movement for combined waves and currents under various ambient conditions and project alternatives. The goal of this type of application would be to provide the technical information necessary to evaluate a placement site that is economically feasible for the optimum sediment placement location.

The CMS model can operate with a non-uniform rectangular or quadtree (telescoping) grid. Quadtree grids combine the advantages of a variable-sized mesh for resolution of complicated geometry with those of an efficient numerical solution to provide a fast, efficient and easily developed modeling approach. CMS-Flow with a quadtree grid uses a fully unstructured approach; all cells are numbered in sequence and pointers are used to determine the connectivity with neighboring cells. The greatest advantage of a quadtree grid over a non-uniform rectangular grid is the freedom to increase or decrease solution resolution where required. Figure 2 shows the quadtree grid of Humboldt Bay with eight different cell sizes. In the quadtree grid, small cells are only used to resolve the geometry where necessary (e.g., around the two jetties) or to provide greater resolution for the flow field (e.g., within the inlet area). The nature of the non-uniform rectangular grid, however, requires that small cells be used along an entire row or column and, consequently, small cells occur in regions where they are not required. More often than not, a quadtree grid has fewer ocean cells than a non-uniform rectangular grid, yet provides better resolution in the areas of interest and greater spatial coverage of the study domain.

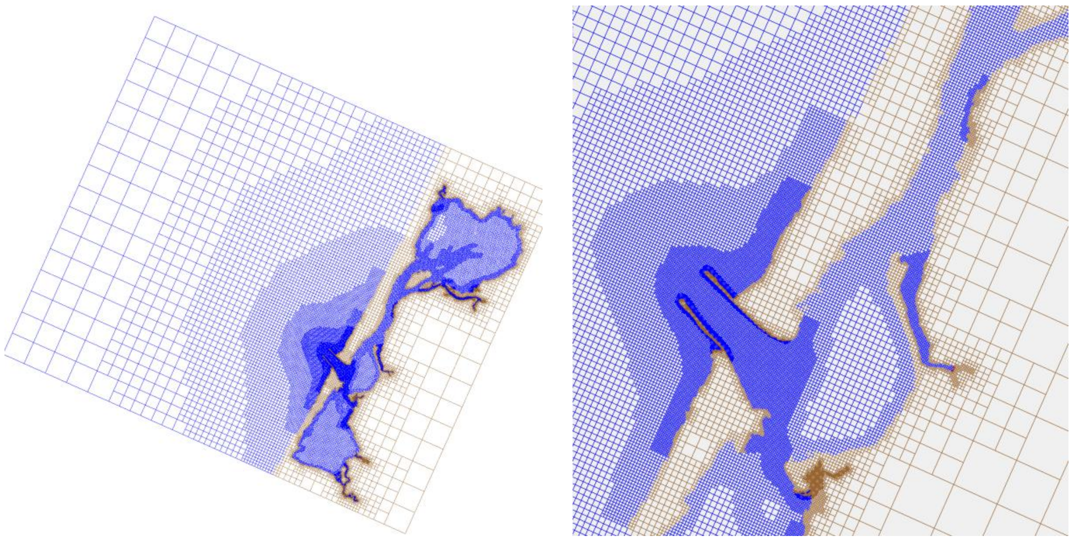

Figure 2. Humboldt Bay CMS quadtree grid.

PTM Setup

The first step in undertaking a CMS/PTM simulation is to load the pre-computed hydrodynamic and wave data into the SMS interface, if it is not already loaded. CMS-Flow requires three input files and CMS-Wave requires six input files that can be developed through the SMS. Each CMS-Flow and CMS-Wave simulation will generate its own output file (Figure 3).

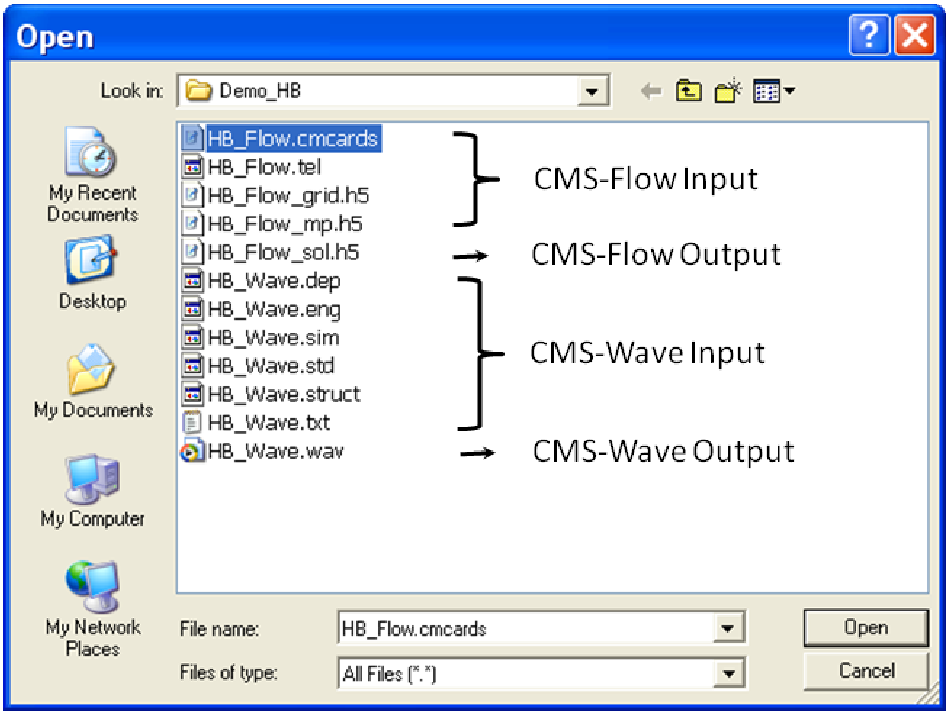

Figure 3. CMS files.

The CMS-Flow files are required, while the CMS-Wave files are optional for setting up a CMS/PTM simulation. After loading the hydrodynamic and/or wave files, users can follow the steps below to set up the simulation for the Humboldt Bay application.

- Particle release sources

  1. Switch to *Map Module* by clicking the icon ,
  2. Right click *default coverage* and choose *Type* | *Models* | *PTM* after generating a point, a line or a polygon for the particle release (Figure 4),
  3. Choose the *Select Feature Polygon* icon  and click on the area source,
  4. Right-click the polygon, select *Attributes*, and *Feature Object Attributes* will pop up,
  5. Select *Horizontal Polygon Source* in the *Type* (Figure 5),
  6. Specify *Date/Time* for the particle release, *Parcel Mass (kg)*, released mass *Rate (kg/m^2/s)*, and *Median Grain Size (mm)* of released particles (Figure 5),
  7. Save the area source specification as a map file.

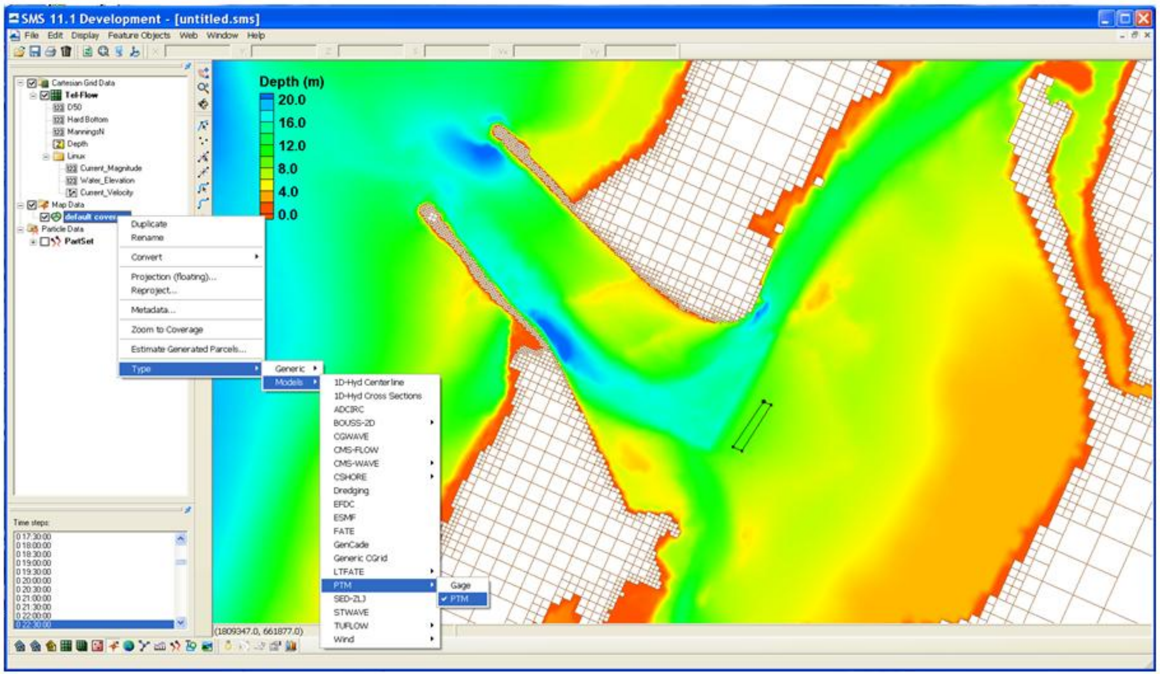

Figure 4. Link source file to CMS-PTM.

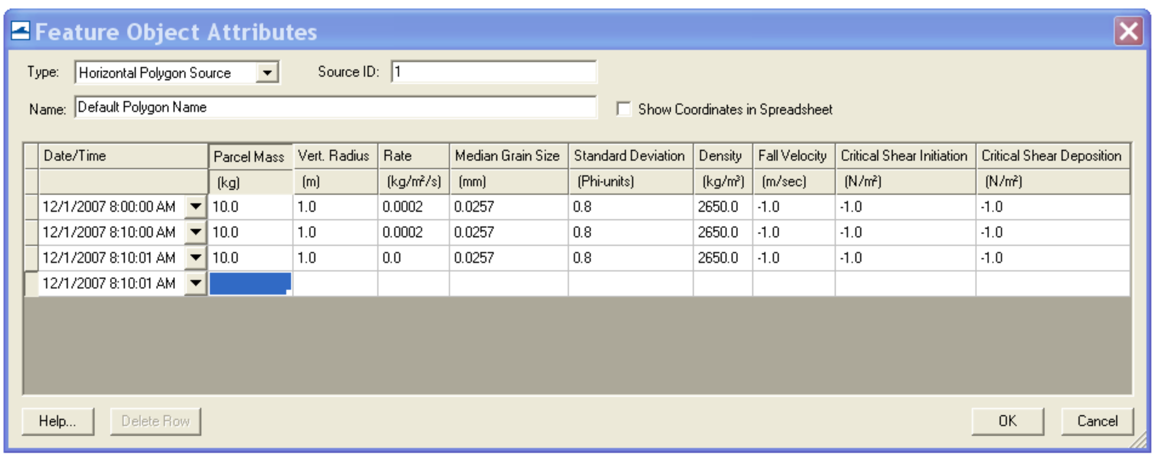

Figure 5. Defining the particle release scenario from the area source.

- CMS/PTM setup

After the hydrodynamic data and source file are generated, users can follow steps below to specify the CMS/PTM parameters and set up a simulation.

1. Switch to *Particle Module* by clicking the icon ,
2. Choose *PTM* | *New Simulation*,
3. Choose *PTM* | *Model Control*.

TIME TAB: The *Time* tab of the PTM interface is the first tab encountered in the model control window (Figure 6). The entries on this tab are used to edit much of the model run-time control. The following sections will discuss each entry.

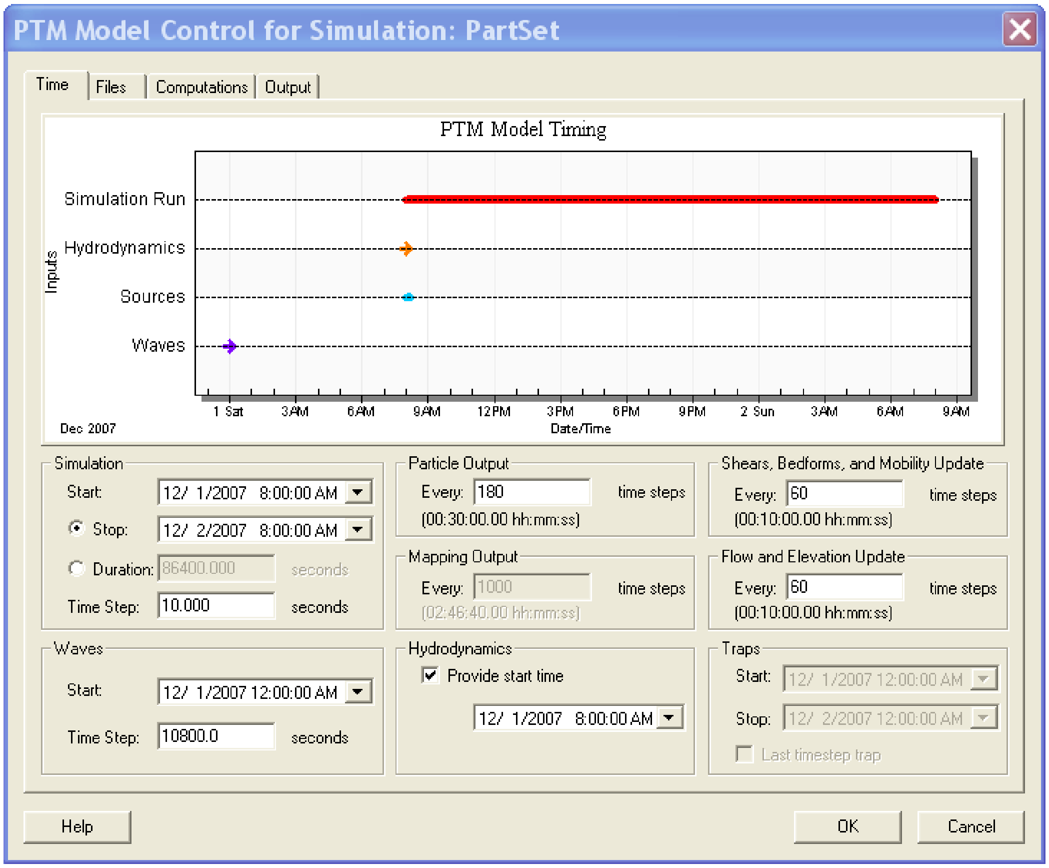

Figure 6. *Time* tab of the PTM control window.

- *Simulation* section

    1. *Start* - The *Start* of the PTM simulation is independent of the driving CMS-Flow simulation, but must be during the time period covered by that simulation.

    2. *Stop* - The *Stop* of the PTM simulation can be set in absolute terms or by selecting a duration.

    3. *Time Step* - The *Time Step* is an important control on the speed and accuracy of the simulation, and should be selected with consideration of a number of factors:

        (a) If CMS/PTM is being used <u>without</u> particles the main constraint in selecting a value for *Time Step* should be the temporal variation of the flow. In a tidal environment, setting *Time Step* to 1800 sec time step and *Mapping Output* to 1 could be used to generate maps of flow and sediment behavior during the simulation. It is important, especially when using this approach, to ensure that the *Shears, Bedforms and Mobility Update* increment is never greater than the *Mapping Output* increment.

(b) If CMS/PTM is being used with particles the value for *Time Step* should reflect the spatial variation of the flow, the geometry and depth of the solution domain and the solution approach used by the model. A longer time step can be used if a 2D advection scheme is selected. For most 2D advection scheme simulations, a 10 sec time step would be acceptable. A shorter time step is required if a 3D advection scheme is selected. For most 3D advection scheme simulations, setting *Time Step* to 1 sec would be acceptable. There are two issues to consider if this approach is used. First, model accuracy can be increased and model run times reduced by decreasing the time step while increasing the *Flow and Elevation Update* increment. Second, the *Numerical Scheme* value (see *Computations Tab* discussion below) should be set to 4, rather than the default value of 2 for highly turbulent flow cases or when particle movement (horizontal or vertical) varies considerably.

- *Waves* section

  1. Identify the *Start* and *Time Step* of the CMS-Wave results file. Data for this section requires knowledge of the contents of the wave results file.

  2. Note that this section will only become editable once waves have been included and a file has been identified on the *Files* tab.

- *Output* section

  1. *Particle Output* - Identify the increment for the output of particle data.

  2. *Mapping Output* - Identify the increment for the output of map data. Outputting this data more frequently than desired can slow down the model's execution and lead to large files. If the flow does not change rapidly, then output can be of the order of many hours. For more rapidly changing simulations output is normally set to be of the order of 30 min to 1 hr. This value should always be greater than or equal to the larger of the two update increments *(Shears, Bedforms, and Mobility Update* and *Flow and Elevation Update*).

  Data to be output is controlled via check boxes on the *Output* tab. The *Mapping Output* will only become editable once a map output box is checked on the *Output* page. The interface displays the frequency in real time units below the option box to assist the user (times are based on the presently selected S*imulation - Time Step*). The optimum output frequency depends on the nature of the flow and desired resolution of the solution.

- *Hydrodynamics* section

  Identify the start time of the hydrodynamics results file, if required. If the box is not checked, then the start time given in the CMS-Flow solution file will be used. Checking the box allows users to overwrite this time. This option is required if the input hydrodynamics file does not contain the start time (most ADCIRC-based simulations).

- *Update* section

  1. *Shears, Bedforms, and Mobility Update* - Identify the increment at which the model re-computes various Eulerian quantities. Even in slowly varying cases (e.g., tidal flows), certain bedforms can develop and wash-out quite rapidly. Standard settings are usually of the order to achieve frequencies of 5 min to 10 min. The update frequency for those variables should always be less than or equal to the increment used for *Mapping Output*.

  2. *Flow and Elevation Update* - Identify the increment at which the model re-interpolates the flow from the CMS-Flow results file to the internal CMS/PTM solution mesh. Setting this update increment value to 30 in a simulation with 2.0 sec time steps for a case in which the flow speed varies by a maximum of 60 cm/sec over an hour, means that the flow used by the model would only be in error by 1 cm/sec. Although in certain cases an updating every time step is justified, standard settings are usually of the order to achieve frequencies of 1 min to 5 min.

  The interface displays the frequency in real time units below each option box to assist the user (times are based on the presently selected S*imulation - Time Step*). The optimum update frequency is important in achieving a fast and accurate solution. This value strongly impacts the run time of a PTM simulation because calculation of these values can be quite time-consuming, especially for a domain with a large number of cells. Updating more frequently than necessary can slow down the model's execution. The user should select this value after consideration of the temporal variation of the flow.

- *Traps*

  This section is used to control the start and stop of particle traps used in the simulation. This capability only becomes available if traps are active. If a trap is to be active for the whole duration, then any *Start* and *Stop Time* that span the simulation time period can be used. If the *Last timestep trap* check-box is checked, then the trap only operates on the final time step of the simulation.

FILES TAB: The *Files* tab of the CMS/PTM interface is used to edit much of the model run-time control (Figure 7). Entries for this tab are discussed next.

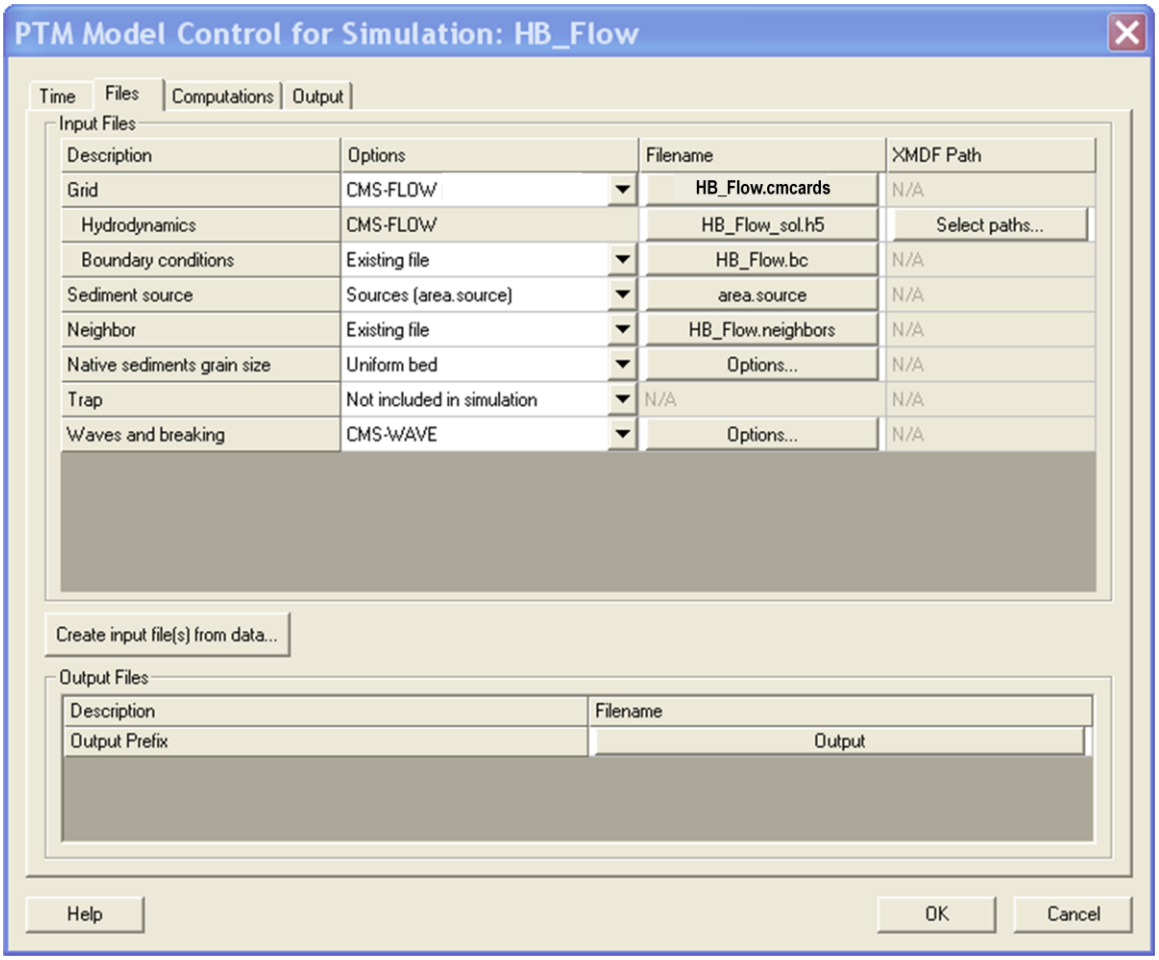

Figure 7. Files tab of the PTM control window.

- *Input Files*

  1. *Grid* - Set to CMS-Flow under the *Options* tab and the name of the cards file (HB_Flow.cmcards for the Humboldt Bay case) should be supplied in the *Filename* column. The PTM will determine the type of simulation (e.g., standard, non-uniform rectangular or quadtree grid) from the contents of the cards file.

  2. *Hydrodynamics* - Link to hydrodynamic results, HB_Flow_sol.h5, under the *Filename.* Click *Select paths* under the *XMDF Path* and Set *Simulation/Current_Velocity* for *Velocity*, and *Simulation/Water_Elevation* for *Water surface elevation*.

  3. *Boundary conditions* – A boundary file, usually given the filename extension "*.bc", is required to allow the model determine what to do at the limits of the CMS-Flow domain. The model automatically generates this file and, if this is the first simulation using this CMS-Flow grid, the *Options* tab should be set to "*Create when model is run*". If a boundary conditions file has been generated by a previous CMS-PTM simulation,

then "*Existing file*" can be selected in the *Options* column and its name entered in the *Filename* column. This file is also required to view PTM map output files.

4. *Sediment source* - Select *default coverage* under the *Options* and name "*.source" under the *Filename*.

5. *Neighbor* - Select "*Create when model is run"* under the *Options* and name "*.neighbors" under the *Filename*.

6. *Native sediments grain size* - Input information about the bed sediments over the domain. Select "*Uniform bed*" from the *Options* column for specification of a single grain type for the entire domain. Click "*Options*" in the *Filename* column. This opens a pop-up window where the three representative grain sizes can be entered (Figure 8). For spatially varying grain sizes, PTM requires native bed sediment data specified as datasets, d35, d50, and d90, in the CMS-Flow grid:

   (a) Select the CMS-Flow dataset under the data tree.

   (b) Choose *Data* | *Data Calculator* in the CMS-Flow menu.

   (c) Name the first dataset, d35, in *Output data set name* and specify a constant value, 0.1 mm in *Calculator*.

   (d) Click *Compute* to generate the dataset.

   (e) Repeat steps (c) and (d) to generate the datasets for d50 and d90 (Figure 9).

   (f) Click *Done* to exit *Data Calculator.*

   (g) Highlight the datasets, d35, d50, and d90, right-click, and select *Export* (Figure 10).

   (h) Select *XMDF file* in *File Type*, check *All time steps* in *Time Steps*, and click *Filename* to save the file NativeSediments.h5 in the present working folder.

7. *Traps* – Link to a trap file and activate traps. The trap file is generated following the same steps for specifying a particle release source.

8. *Waves and breaking* - Select *CMS-Wave* under the *Options* and click *Options…* under the *Filename* to obtain wave input for PTM (Figure 11).

   (a) Click the icon  to browse the working folder and to link the wave simulation file, HB_Wave.sim, under *Parent Grid Geometry* | *CMS simulation file*.

(b) Click the icon  to link the wave input file, HB_Wave.wav, under *Parent Grid Solution Data*.

(c) Specify *Number of Spectra in file*.

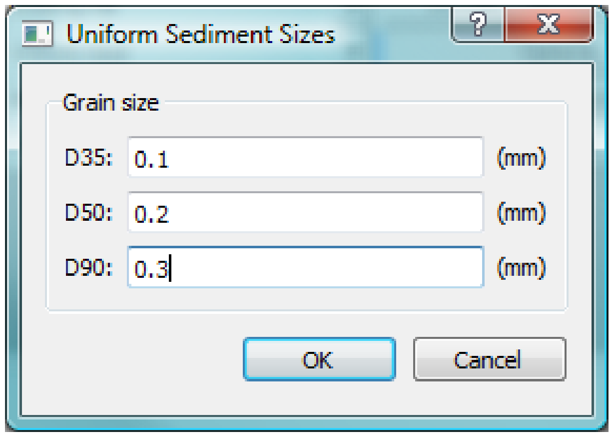

Figure 8. Pop-up window used to identify uniform bed grain size distribution.

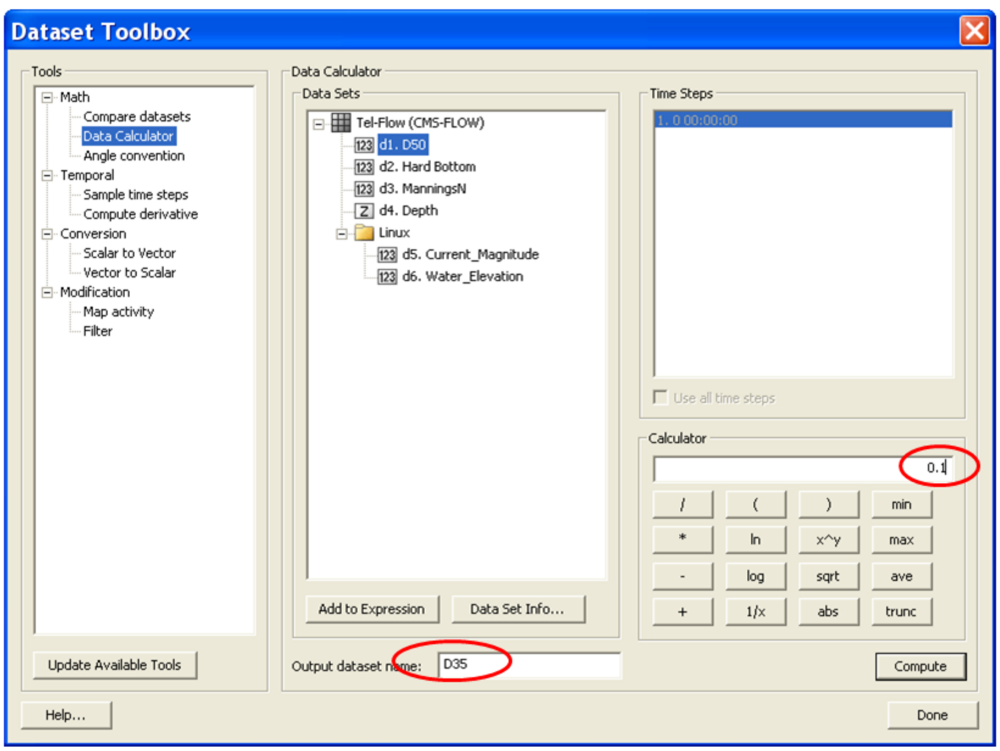

Figure 9. Generating spatially-varying sediment datasets, d35, d50, and d90.

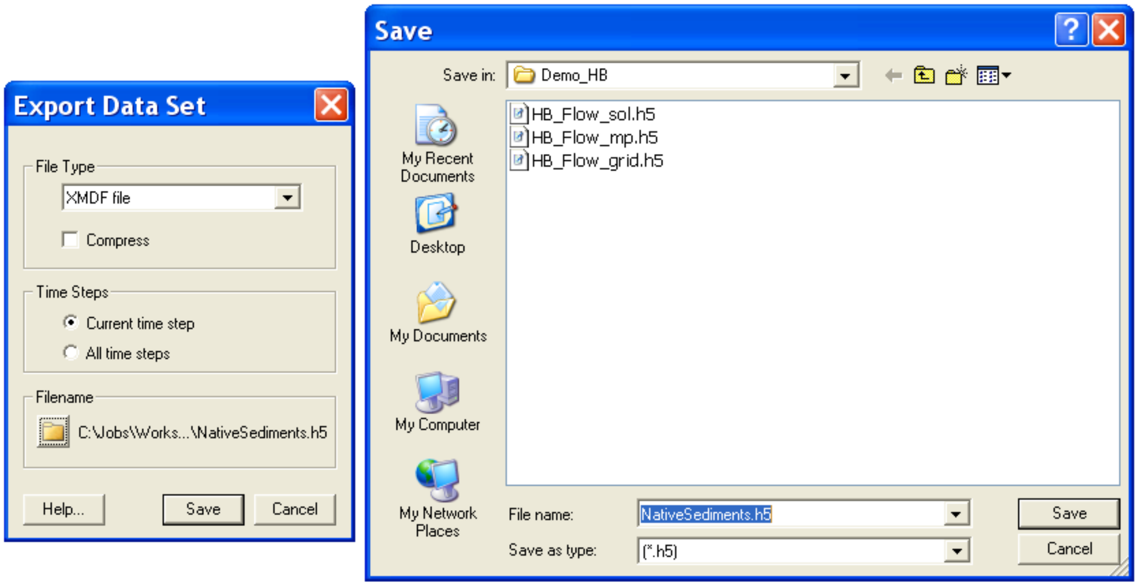

Figure 10. Generate the native sediment file.

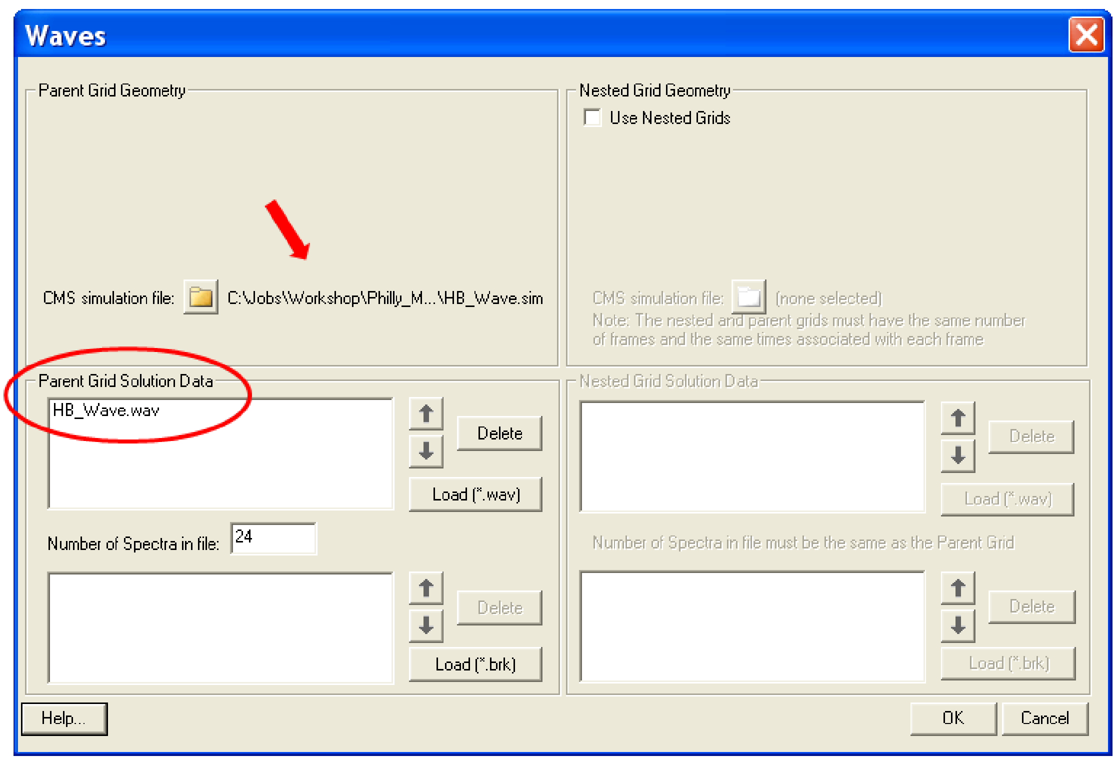

Figure 11. Specifications of wave input.

Because the wave simulation is included in CMS-PTM, it is necessary to return to the previous (*Time*) tab to specify the *Start* time and *Time Step* under *Waves* (Figure 6).

COMPUTATIONS TAB: The *Computations* tab of the PTM interface is used to edit model run-time settings (Figure 12). Each entry is discussed in the next section.

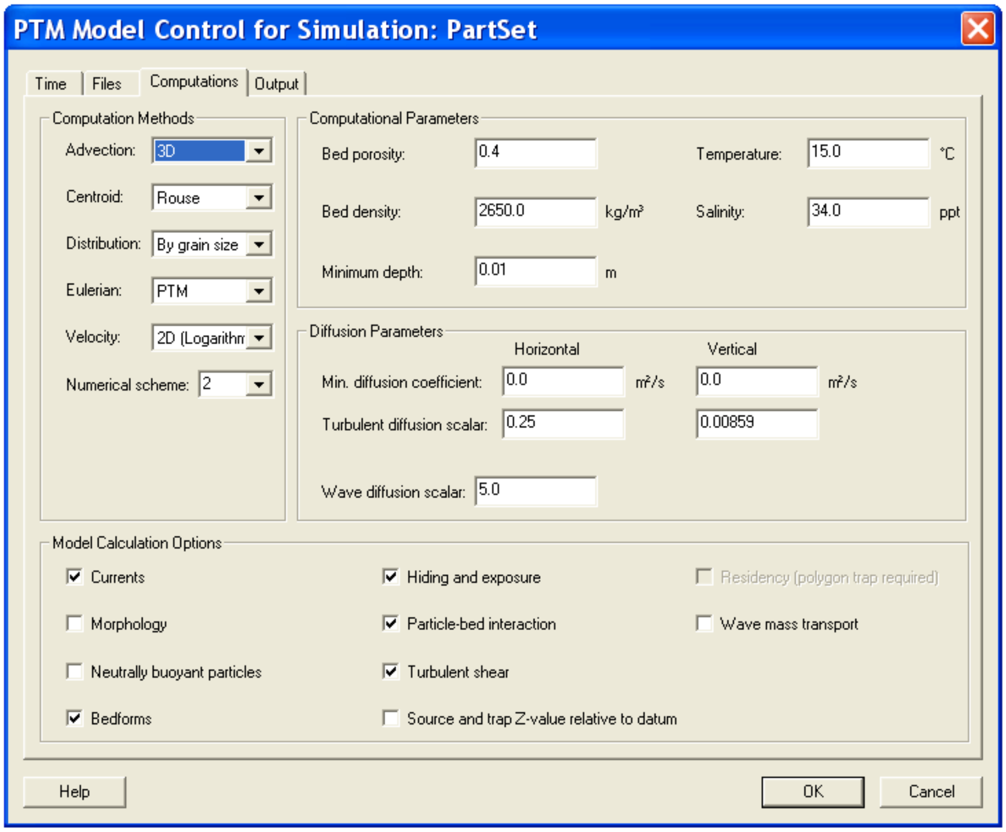

Figure 12. Computations tab of the PTM control window.

- *Computation Methods*

1. *Advection* - Define the solution approach for advection of individual particles. The options available are *2D* and *3D*. The 2D advection scheme does not advect particles vertically, but places them at the vertical centroid of the local sediment transport distribution. This method allows longer time steps to be used. It is important that an appropriate value for the *Time Step* is selected based on whether 2D or 3D is selected.

2. *Centroid* - Determine the elevation of the centroid of the sediment transport distribution above the bed if a 2D advection simulation is being used. The default setting, *Rouse*, will result in much faster simulations than the *Van Rijn* option and is recommended in all cases.

3. *Distribution* - Control whether particle sources are constructed with particles by their grain size or weight. The default setting (*grain size*) is appropriate for most cases, but if a specific particle size distribution based on "*percent passing/retained by weight*" is being sought then the alternative option should be selected. In this case, a spherical approximation is used to convert diameter to weight.

4. *Eulerian* - Control the techniques to compute much of the background calculations (e.g., shears, mobility, etc.) over the solution domain. The default setting, *PTM*, will result in much faster simulations than the *Van Rijn* option and is recommended in all cases.

5. *Velocity* - Determines the vertical distribution of the input flow field. In the present case, the "*2D (Logarithmic)*" option should be selected for the CMS-Flow hydrodynamic model input.

6. *Numerical scheme* - Choose between a second or fourth-order Runge Kutta scheme. The fourth-order, while slightly slower for simulation with large numbers of particles, is more accurate.

- *Computational Parameters* - Describe the characteristics of the native bed sediment and water column. The *Minimum depth* value also controls the minimum depth for particle transport.

- *Diffusion Parameters* - The horizontal and vertical *Turbulent diffusion scalar* values are used to scale the random walk routine. These calculations are based on the input flow field. If it is believed that the flow is more diffusive than the input flow field would suggest (e.g., sub-grid scale processes, high frequency flows from winds or waves, etc.), then minimum values for the diffusion coefficients can be specified. Non-zero values should be used with care and should be tested to ensure that they are appropriate for the specific application.

- *Model Calculation Options*

1. *Currents* - Read and use an input flow field. This should always be checked.

2. *Morphology* - Compute potential changes to the depths over the solution domain. These calculations are based on the Eulerian calculations and not on the particle results.

3. *Neutrally-buoyant particles* - Treat all particles as if they were neutrally buoyant. The *Advection* scheme should be set to *3D* for this option. Particles should not deposit if this option is used; however, particles could still reach the bed by diffusion, especially if a large time step is used.

4. *Bedforms* - Compute bedform dimensions for the native sediment and flow conditions. This option should be disabled if the bed is known not to have bedforms.

5. *Hiding and exposure*, *Particle-bed interaction* and *Turbulent shear* - These add complexity to the particle transport. The first two depend on the native bed grain size. They should all be used for most applications.

6. *Source and trap Z-values relative to datum* - Control the vertical datum of <u>all</u> sources and traps.

7. *Wave mass transport* – Causes particles near the bed to move with the waves and those above the bed to move against the waves. Wave data must be input. This option is intended for particles in intermediate water. It should not be used if particles near the surfzone, where it may produce misleading results.

OUTPUT TAB: Identify the model's output (Figure 13). The design of this tab has not changed for this version of PTM that works with a quadtree grid (PTM 2.1).

The PTM uses an unstructured triangular mesh to perform its calculations. In earlier versions of the PTM, this mesh was constructed with mesh nodes located at the centers of the CMS-Flow cells. This approach resulted in a small region around the outside of the CMS-Flow domain being left out of the PTM domain. PTM v2.1 is using the cell corners as the mesh node locations, allowing the full CMS-Flow domain to be modeled in the PTM. The map output of a CMS-Flow application that has been modeled with earlier versions of PTM will thus be slightly different from those produced by PTM v2.1. Due to this change, the user must open this PTM solution mesh to view any output files. In this tab, select *Print default keywords*, *Source*, and *State*.

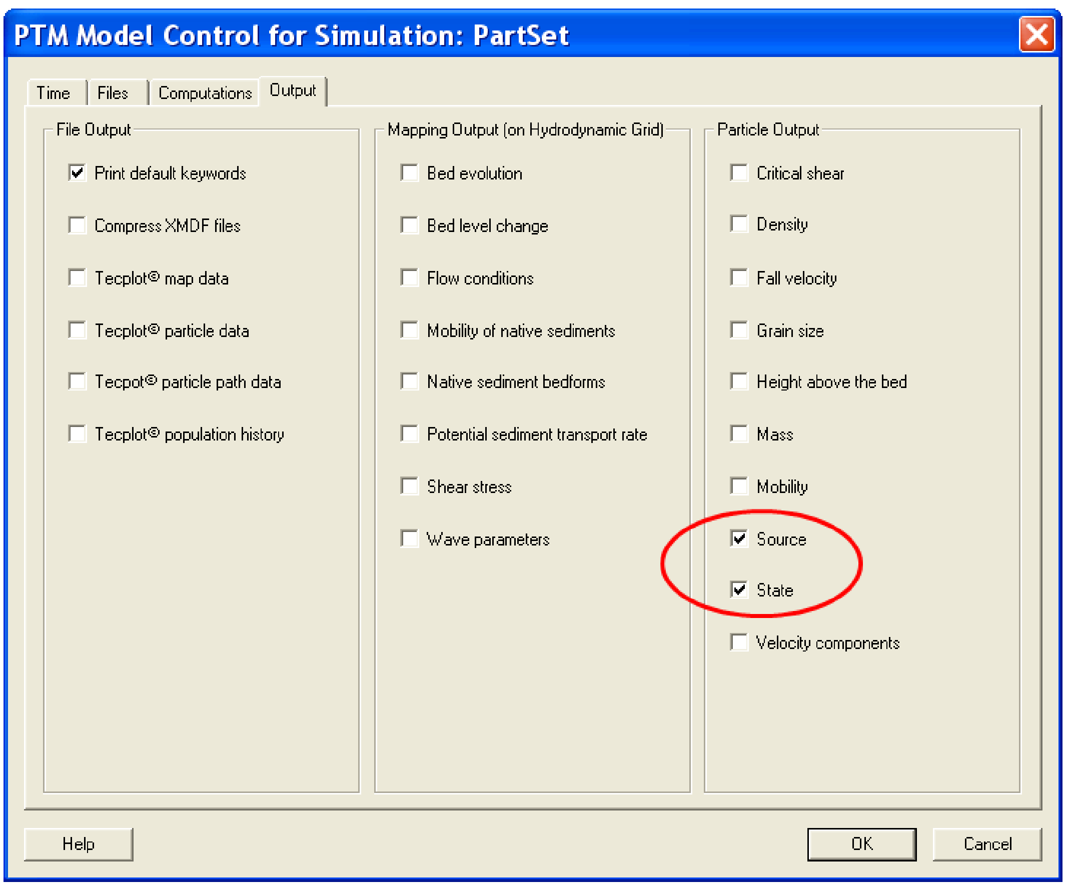

Figure 13. *Output* tab of the CMS-PTM control window.

MAP FILE VISUALIZATION: The *Boundary conditions* file is automatically generated by CMS/PTM with the filename extension ".bc". It is used to allow the model to determine what to do with particles at the limits of the CMS-Flow domain, but is also required to view CMS-PTM map output files. The bc file contains all the data (node location, element connectivity and boundary nodestrings) for a three-noded, triangular finite element mesh. To view a PTM map output file, first open the boundary conditions file using the *File | Open* command. The file will appear as "Mesh" with a single elevation data set associated with it. After this file is opened, the map file can be opened and visualized. Figure 14 shows the SMS data tree for a simple CMS-PTM simulation before and after the map file is opened. Note that SMS associates the map file output with the Mesh object, instead of the CMS-Flow grid.

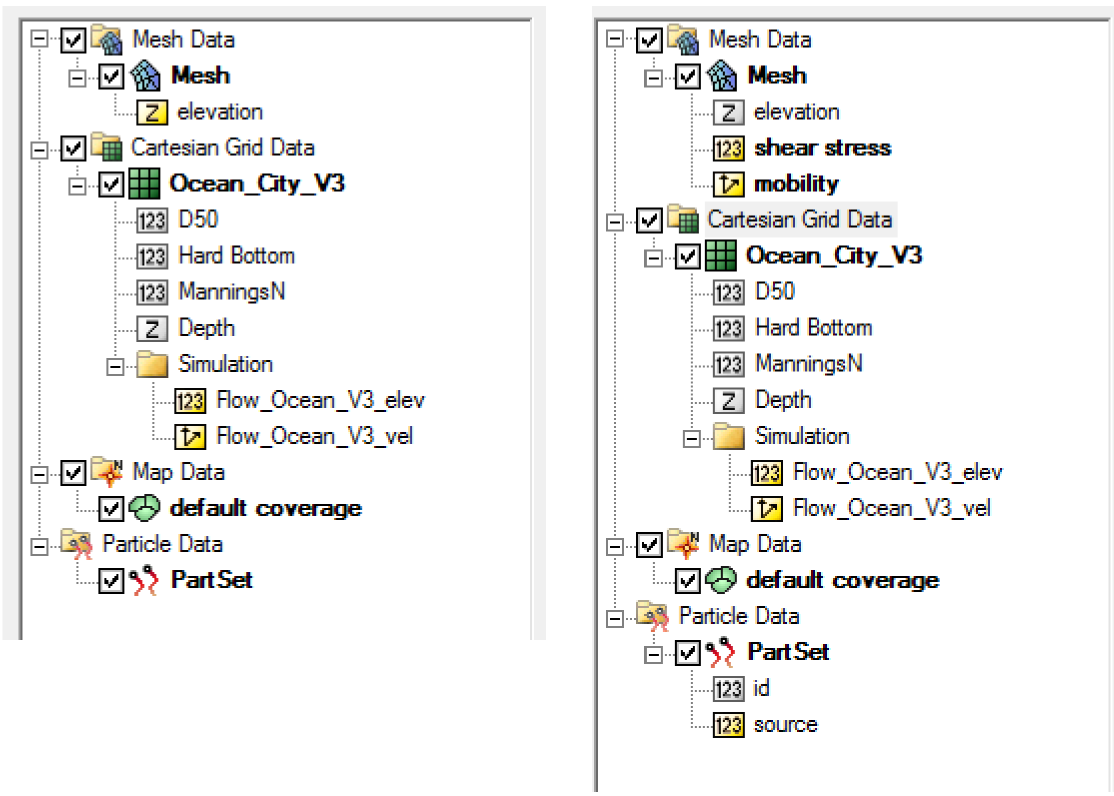

Figure 14. Example data tree: after opening bc file (left); after opening map and particle files (right).

Run PTM

Once the input on the various control pages is completed, the PTM can be checked (*PTM | Model Check...*) and executed (*PTM | Run Model*). For the first simulation of a model setup, the PTM will generate boundary nodestrings. These may be checked to ensure that the open/closed settings are correct. By opening the *.bc* file and selecting the nodestring(s), the boundary condition setting of one or more nodestrings can be changed. The boundary type *Ocean* is used for open boundaries and *Mainland* for closed boundaries.

PTM Results

The following example presents the results of a continuous release of 0.1 mm sediment from a line source in the inlet throat (Figure 15). The simulation covers 24 hours, although the source is set up for a longer period (Figure 16).

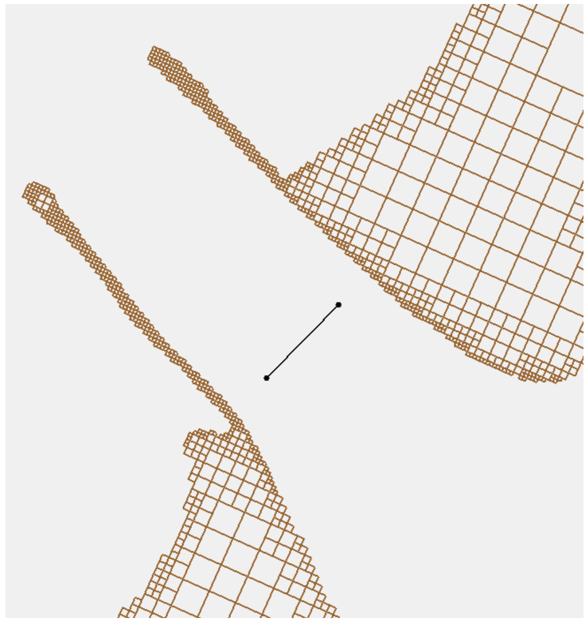

Figure 15. Line source location.

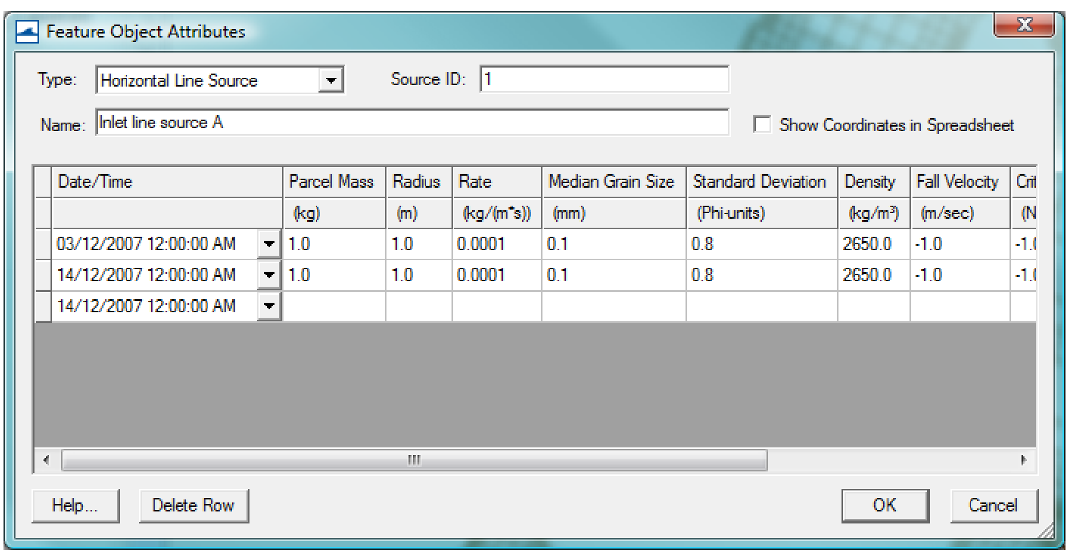

Figure 16. Line source: location release details.

The results of the simulation are shown in 6-hourly increments in Figures 17 through 24. The plots to the left show the particles' grain size, *D*, and those on the right show the particles' mobility, *M* (a particle becomes immobile if $M < 1$). The early stages of the simulation cover a period of ebb tide and the particles are mostly seaward of the source location. As expected, the finer particles have traveled further (Figure 17) and only those in the throat of the inlet are mobile at this time (Figure 18). Figures 19 and 20 show the particles at the start of the flood tide. As flows in the inlet increase, more of the particles become active. By 18 hours into the simulation (Figures 21 and 22), the ebb tide has started again, mobilizing almost the entire inlet area. At 24 hours (Figures 23 and 24), the particles have been dispersed over a large area both inside and outside of the inlet. As illustrated earlier, the larger particles remain close to the release area while the smaller particles travel the furthest. Particles transported the furthest away from the inlet throat tend to remain immobile throughout the entire tidal cycle and thus illustrate the initiation of long-term formation of flood and ebb shoals.

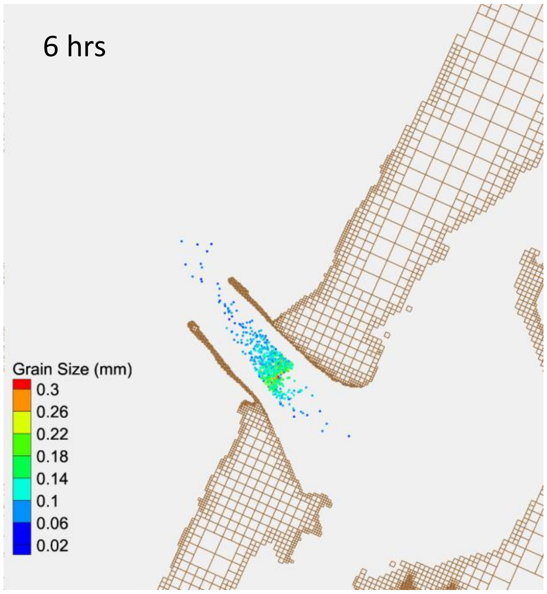

Figure 17. Particle distribution (grain size).

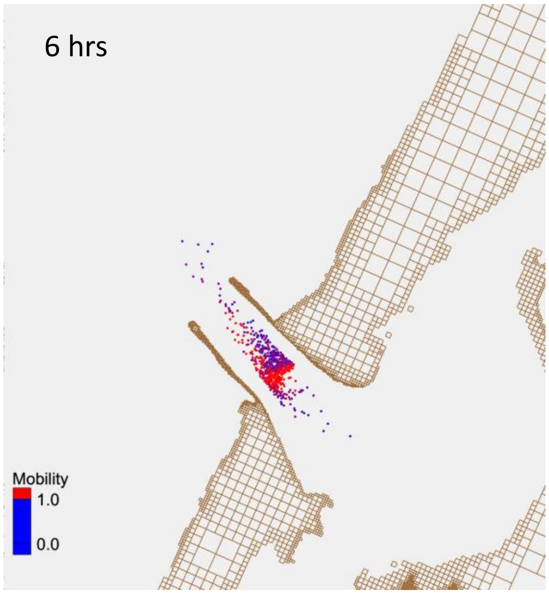

Figure 18. Particle distribution (mobility).

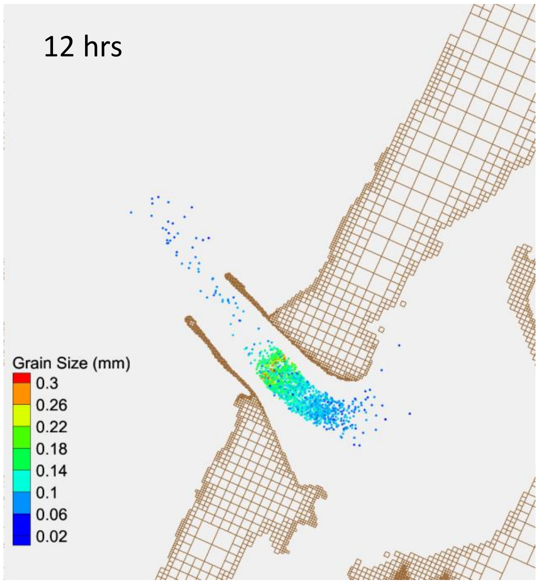

Figure 19. Particle distribution (grain size).

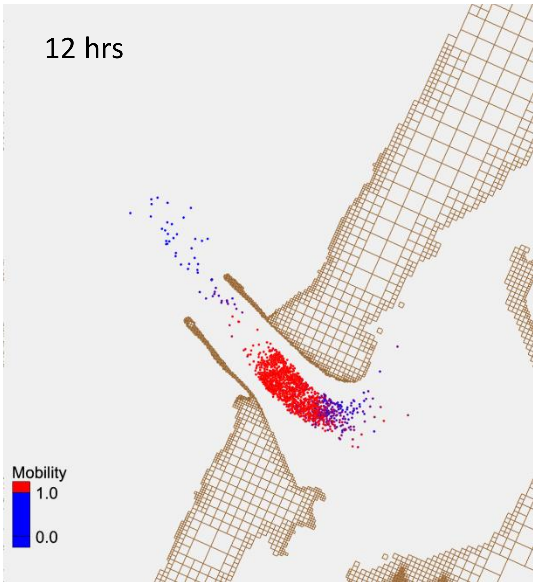

Figure 20. Particle distribution (mobility).

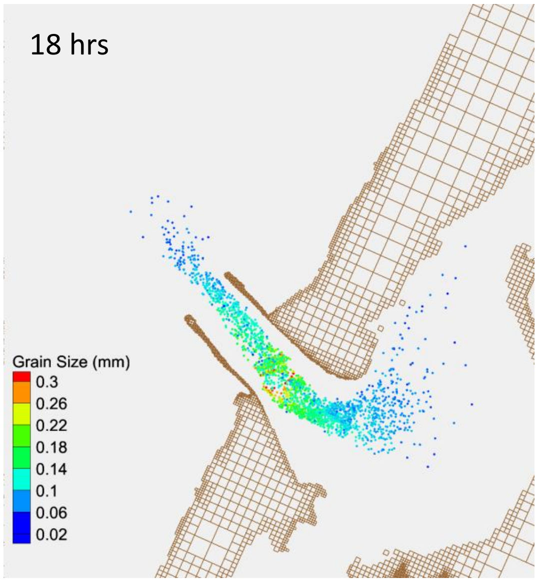

Figure 21. Particle distribution (grain size).

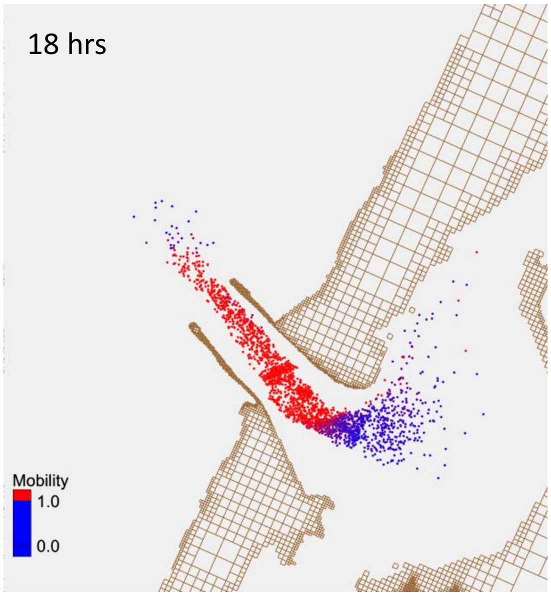

Figure 22. Particle distribution (mobility).

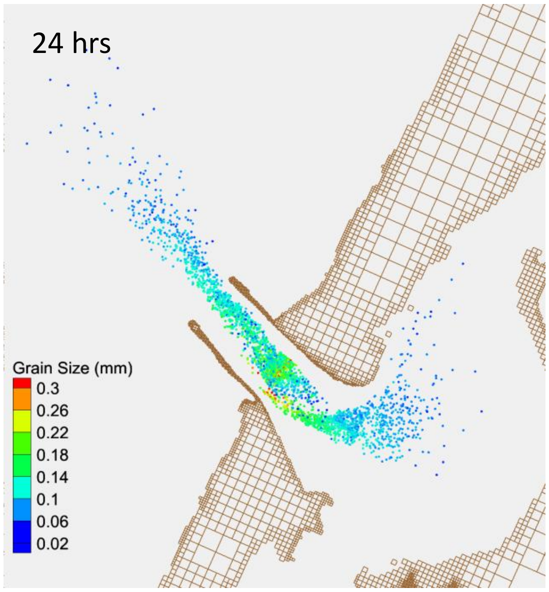

Figure 23. Particle distribution (grain size).

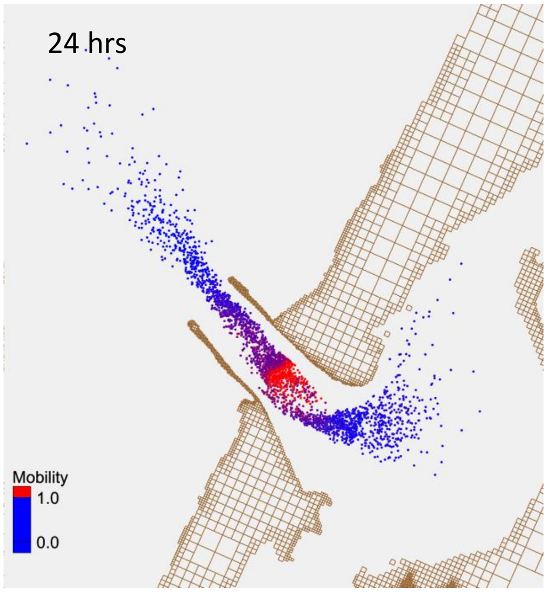

Figure 24. Particle distribution (mobility).