



Particle Tracking Model (PTM) in the SMS: III. Tutorial with Examples

PURPOSE: This Dredging Operations and Engineering Research (DOER) Technical Note (TN) is a tutorial with examples of the PTM, developed jointly by the Coastal Inlets Research Program (CIRP) and DOER Program. This note is applicable to Version 1.0 of PTM. Demirbilek et al. (2005a) describe the PTM interface, and an overview of features and capabilities of the PTM is presented in Demirbilek et al. (2005b). The theoretical formulation and implementation of the PTM are given in a technical report (MacDonald and Davies, in preparation).

INTRODUCTION: The PTM is a Lagrangian particle-tracking model that is part of the U.S. Army Corps of Engineers (USACE) Surface Water Modeling System, SMS (Zundel 2005, Zundel et al. 1998). It employs a Lagrangian method of tracking particle pathways to estimate migration of sediment particles as influenced by waves and currents. For its input, the PTM requires a geometric surface defining the bottom elevation (depth) over which water level, current velocity vectors, and waves are available at each point in the modeling domain. The user specifies sediment sources and model parameters to perform a PTM simulation within the SMS for a given set of hydrodynamic input (waves, water levels, and currents). The SMS includes commands for layout of the sediment sources, specification of the numerical parameters, and management of the Eulerian quantities (water depth, surface elevation, current velocity).

This document describes typical steps and methodologies users can employ when performing a PTM simulation. The water level, velocity, and wave fields must be simulated or specified prior to constructing a PTM simulation in SMS. This document provides instructions for constructing a PTM simulation, including definition of particle sources, specification of model options and parameters, and coordination of timing for various inputs to the simulation. Commentary is provided to explain the underlying methodologies and reasoning for each step.

LOADING EULERIAN DATA: Eulerian data define the input bathymetry and domain boundary definition as well as input wave and current conditions. The PTM reads these from input files. Wave and hydrodynamic analyses must be performed prior to the PTM simulation. At present, the PTM requires a computation domain defined in the form of an Inlet Modeling System (IMS) -ADCIRC (Luettich et al. 1992) grid file, with an XMDF (Jones et al. 2004) data file containing water surface elevations and currents. If wave input is desired, STWAVE (Smith et al. 2001) is the default wave model, and the user can provide PTM an STWAVE grid and the associated wave field file. The user can also use SMS to convert result files from other hydrodynamic or wave models to these required formats prior to using them in a PTM application.

The example simulation files used in this tutorial come from an analysis performed for a hypothetical generic estuary. Required PTM input files from the IMS-ADCIRC and STWAVE

simulations are provided as part of the standard tutorials distributed with the SMS (version 9.0) package. These files are also available for downloading at <http://el.erd.c.usace.army.mil/dots/doer/doer.html>.

The grid file (estuary.grd in fort.14 format) defines the domain of the PTM simulation. It includes the coordinate location and bathymetric elevation for each node in the grid. The XMDF format hydrodynamic file (estuary_tide.h5) includes water surface elevations and current values for each node at 30-min intervals. These water levels and currents are required input for the PTM. After launching the SMS, the first step in creating a PTM simulation is to select and load the grid file, thus providing a framework for all other Eulerian inputs and outputs. To do this, go to the *File/Open* and select the file “estuary.grd” in SMS. Repeat the command to select “estuary_tide.h5” to load the hydrodynamic solution. The data will be displayed in SMS as shown in Figure 1.

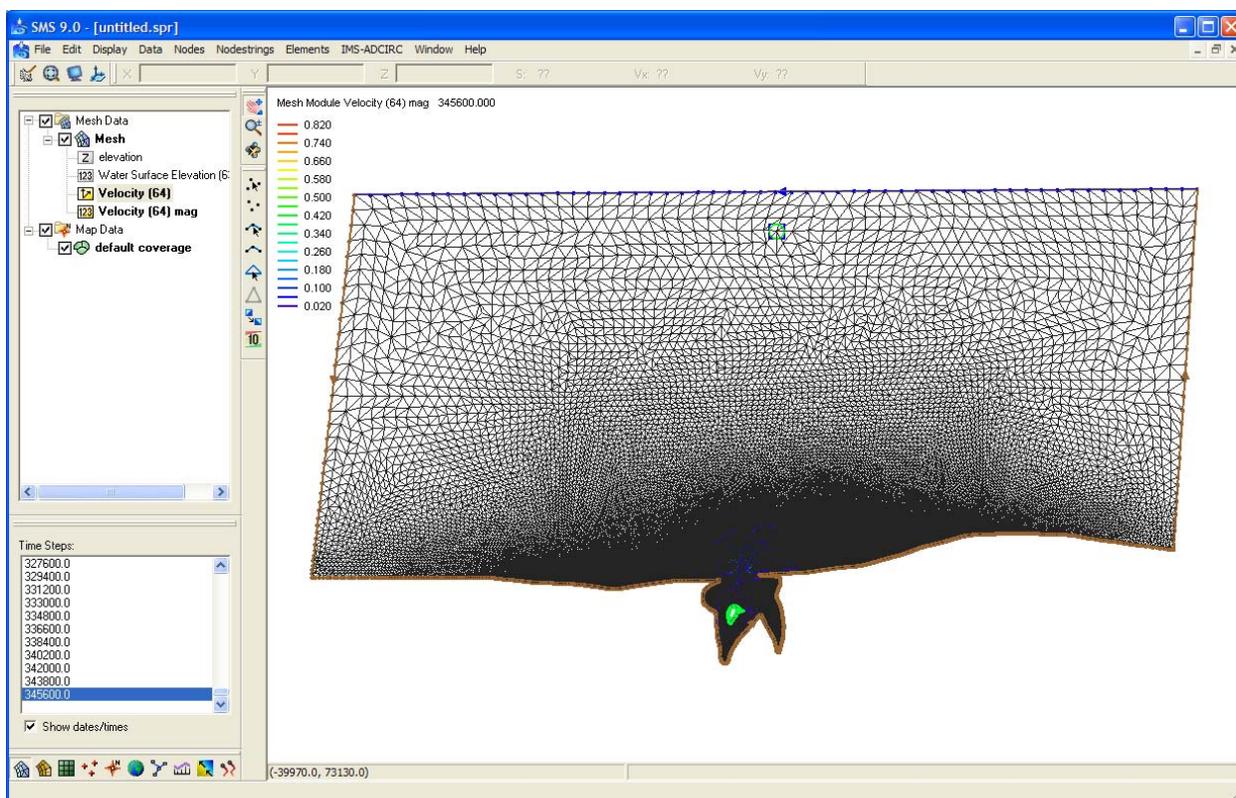


Figure 1. Hydrodynamic mesh and solution data sets visible in the *Project Explorer* window

ADCIRC output files can be large. Therefore, it may be advantageous to trim spatial and temporal range of ADCIRC output to create PTM hydrodynamic input files. This can be performed from within SMS (Zundel 2005). Care must be taken not to spatially trim too much of the grid because once particles reach the edge of the domain, they are lost to the simulation. Therefore, the trimming of ADCIRC output should not result in particles leaving the domain unless the flow patterns could keep the particles from returning to the area of interest.

The PTM interpolates the hydrodynamic solution data to the Lagrangian particle locations. Therefore, the locations of the ADCIRC nodes are not important for display. In fact, they tend to clutter up the image, so it may be useful to turn them off, by selecting the *Display Options*  macro and turning off the *Elements* toggle. Make sure the *Nodes* toggle is also off. Turn on contours (the *Contours* toggle) and the velocity vectors (*Vectors* toggle). Next click on the *Vectors* tab of the *Display Options* dialog and select the option to display vectors on a grid, and click the *OK* button. The vectors that appear on the screen illustrate the nature of the flow field for the selected time-step. Note that at the first time-step, there is little circulation due to the ramping of the forcing condition. Users may also need to adjust the settings for vector display. The image shown in Figure 2 displays vectors on a 20- by 20-pixel grid with minimum and maximum vector lengths of 10 and 80, respectively.

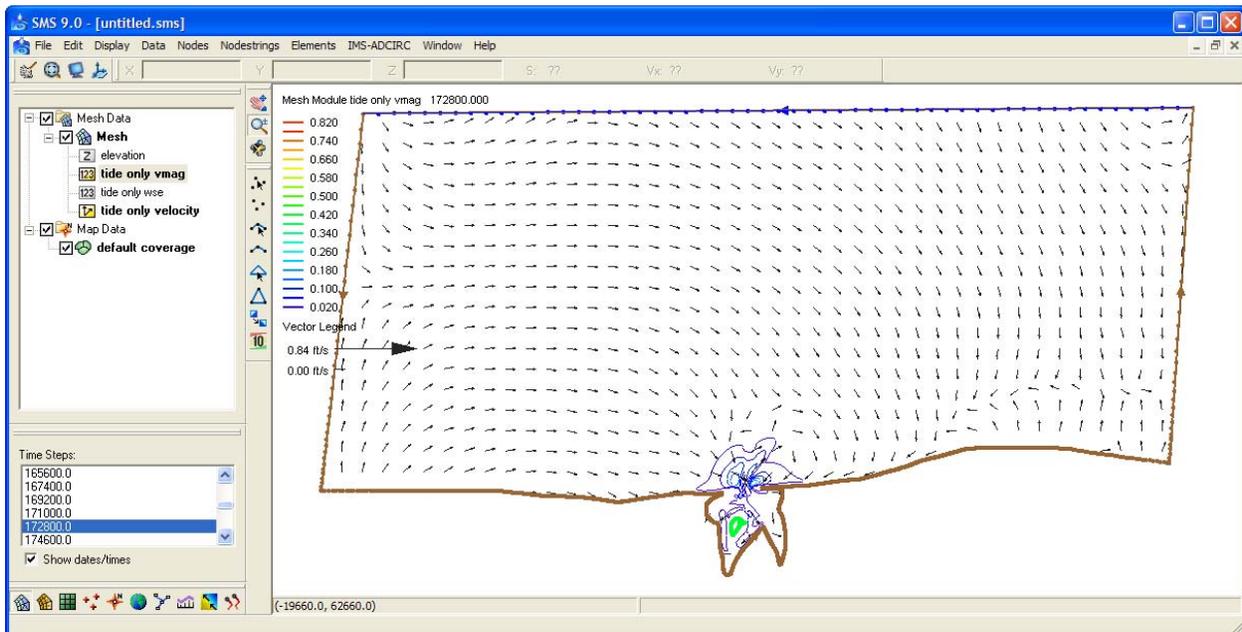


Figure 2. View after hiding ADCIRC grid and showing vectors

If the modeler is not familiar with the hydrodynamics that have been computed for the domain, he/she may want to examine the flow data that are driving the PTM simulation by selecting the data sets from the *Project Explorer* (tree on the left side of the SMS screen). This PTM simulation will use the “tide only velocity” data set, and the simulation will start at the time value 172,800 sec. (explained later). To view currents that will be used by the PTM, click on “tide only velocity” in the *Project Explorer* to select it as the active vector set and click on “tide only vmag” to select it as the active scalar set. Then choose 172,800 as the active time-step (below the tree). Figure 2 shows the view that will result. Note that the names “tide only vmag” and “tide only velocity” in the *Project Explorer* panel on the left are bold. This indicates that they are the data sets displayed in the image.

This simulation will examine particle movement from the coastline just east of the mouth of the estuary. Figure 3 shows a zoomed image of the portion of the domain involved in the Lagrangian calculations. Zoom into this portion of the display using the *Zoom* tool .

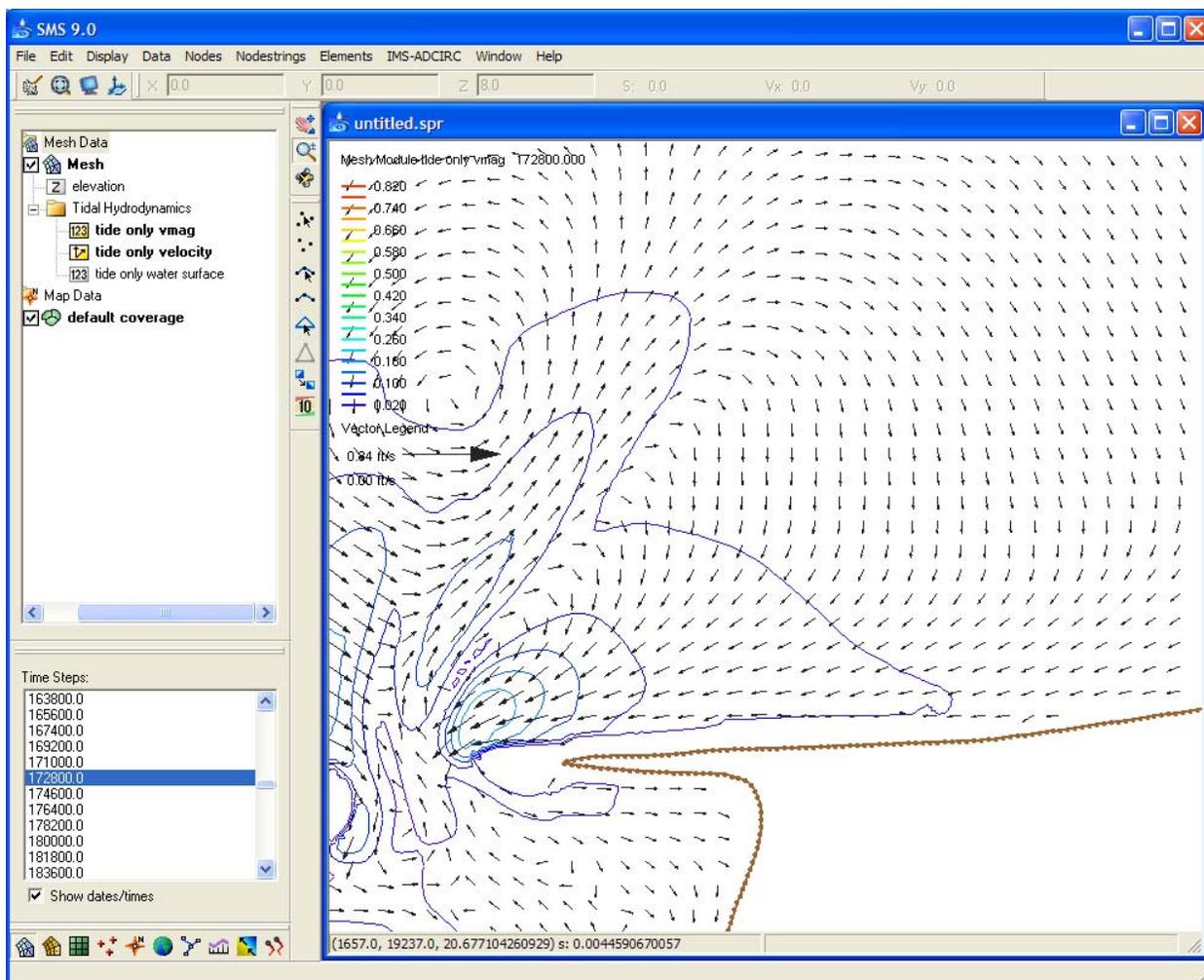


Figure 3. Zoomed view after hiding ADCIRC grid and showing velocity vectors

ADCIRC time series output includes a time stamp (in seconds) for each output interval. These time values are referenced to a starting date, which has been used to reference tidal constituents. Since this example does not represent an actual location, a reference date has been created. Therefore, one can say the time-steps are referenced to 9 February 2004 at midnight (12:00 am). This date corresponds to time zero (0) in the hydrodynamics. The solution file (“*.h5”) begins at the time-step value of 1,800 sec. This corresponds to 12:30 am on 9 February 2004. The first 24 hr of solution include the ADCIRC ramp. Over these time-steps, ADCIRC increases the tidal forcing function from zero to the true forcing. For that reason, the first day should not be used in the PTM simulation; the first two days will be skipped and the PTM simulation will start at the time value of 172,800 sec (2 days), which corresponds to midnight on 11 February 2004. Just over two days of the data will be used in this PTM simulation.

A PTM simulation requires specifying the actual date and time values for the simulation. Because ADCIRC data are referenced to a relative start time ($t=0$), it is necessary to specify the hydrodynamic date/time stamp to align it with the PTM time. Table 1 displays the schedule for the events that will be included in the simulation. These include the hydrodynamics, waves, and

simulated sediment particle sources. The PTM calculations start after the first hydrodynamic output time-step.

Table 1 Chronology of PTM Simulation			
Time	Date	Seconds	Event
12:00 am	9 Feb 2004	0	Hydrodynamic reference time (0 sec in ADCIRC)
12:00 am	9 Feb 2004	0	Beginning time for wave simulation data
12:00 am	10 Feb 2004	86,400	End of ramping calculations
12:00 am	11 Feb 2004	172,800	PTM Simulation Starts – Instantaneous Particle Source Emits Particles
12:00 am	13 Feb 2004	345,600	End of PTM and ADCIRC Simulation

DEFINE THE SEDIMENT PARTICLE SOURCES: In this example, a short-duration line source is used to represent nearshore placement of dredged material. An SMS feature arc (or point) is used to define the PTM source location. To create a PTM source using an SMS feature arc, a PTM-type coverage must first be defined as follows:

- In the *Project Explorer* (tree window in the left panel) right-click on the “default coverage.” Select “Rename” and then type “Placement Option 1.”
- Right-click on the coverage again and move the cursor to the “Type” item. A pop-up menu will appear. Select “PTM” as the type.
- Left-click on the coverage to make it active and to activate the Map module.

Create a feature arc by selecting the *Create Arc* tool . This tool bar is on the panel on the left side of the SMS graphics window. Click in the mouth of the estuary (e.g., the southern left point shown in Figure 4) and then double-click at the right point. SMS creates an arc between the two selected points. These steps are illustrated in Figure 4.

The PTM is used in this example to predict pathways and fate of material placed along this line. The exact positions shown are not crucial. It is important to place the source inside the computational domain so that the generated particles are in water. For reference, the coordinates selected in this example were (x=-4,000 m y=11,700 m) and (x=950 m y=12,000 m). The specified line could represent a proposed placement location for material dredged from the mouth of the estuary. In this case, the line varies from the high-velocity inlet to the calmer shore to illustrate the variation in particle activity.

After specifying the location of the source, it is necessary to set the point source attributes. To do this, go to the *Select Feature Arc*  tool, click on the feature arc, and choose the *Feature Objects/Attributes* command. This displays the *Line Source Properties* dialog (Figure 5). Each row of the dialog represents a change over time in the source. Each column defines an attribute of those particles.

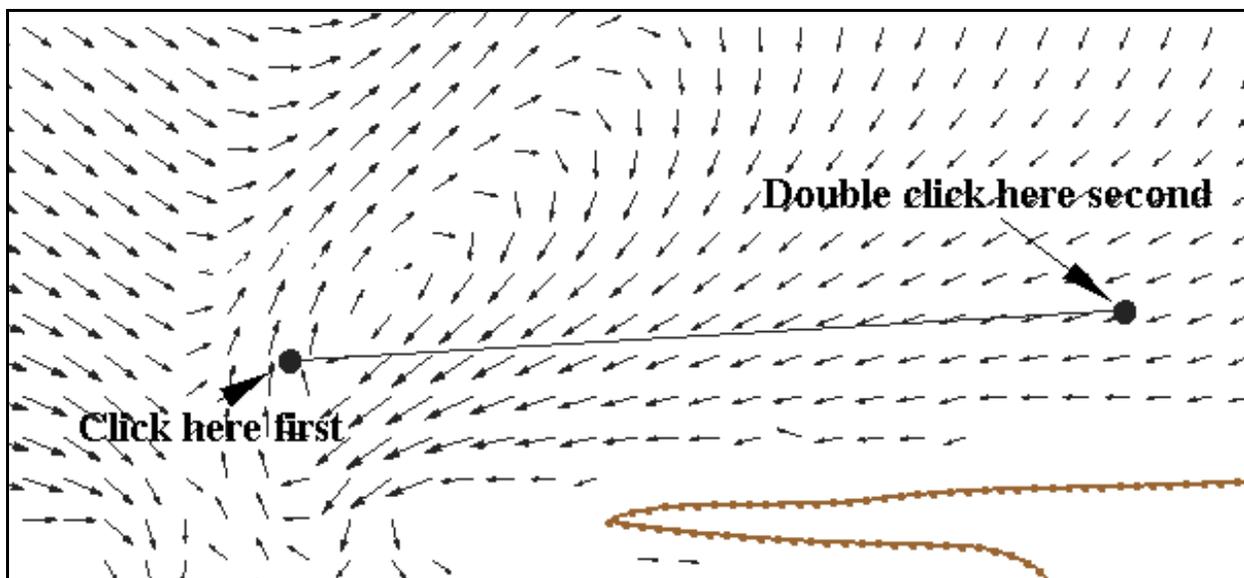


Figure 4. Location of the particle source

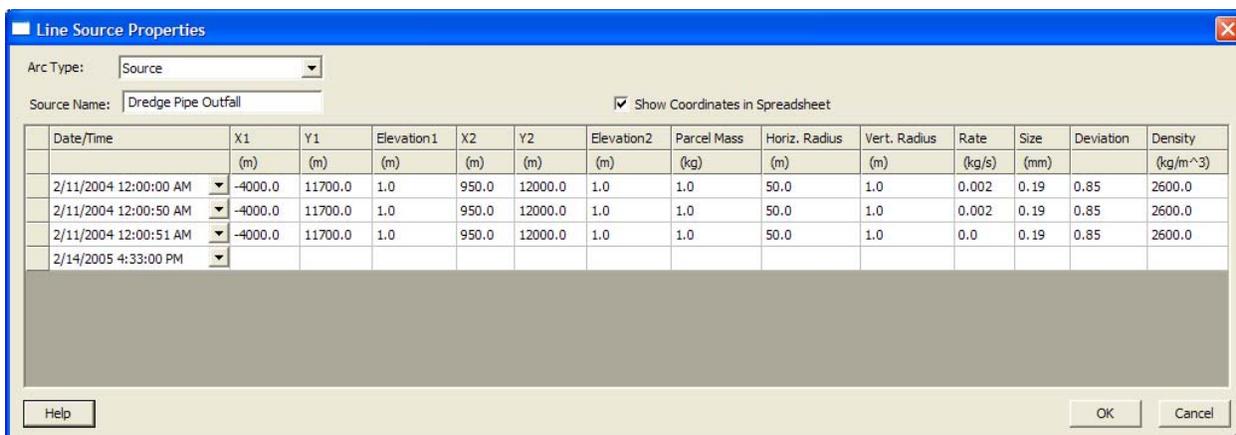


Figure 5. Settings for line source

The first line in Figure 5 defines the time when the source begins to generate particles. Double-clicking on one of the date or time entries and then using the left and right arrow keys in the first column allows the user to enter 2/11/2004 12:00:00 AM as shown in Figure 5. The second, third, fifth, and sixth columns show the coordinates of the segment endpoints. The particles generated are centered around this line. The user can edit these locations, and change the location of the feature arc in this spreadsheet. Enter the values specified above to move the feature arc to the specified location. By changing the location in different rows, the location of the particle source can move over time. The fourth and seventh columns specify the centroidal elevation; enter a value of 1.0 to indicate that material will be released 1.0 m above the bed. The eighth column is the mass of each particle tracked by the PTM. In actuality, this particle is a cluster of particles. Each particle cluster has a characteristic grain size that controls how the particle acts. The mass entered in column 8 helps to account for more material. Enter a value of 1.0 to tell the PTM to create particle clusters of 1.0 kg mass.

The ninth column specifies the radius (enter 50.0 m) in the horizontal plane to distribute particles, using a Gaussian distribution type. The tenth column specifies the vertical distance (enter 1.0 m) to distribute particles using a linear distribution. The eleventh column controls the number of particles (or particle clusters). It allows the user to specify the rate of mass of the material to be released per meter per second (enter 0.002 kg/m/sec, which yields 0.002*5000 m (length of horizontal line source) or 10.0 particles/sec, each with a mass of 1.0 kg). The twelfth column is the characteristic particle grain size in millimeters, and in this case, $d_{50} = 0.19$ mm. The thirteenth column specifies the standard deviation value of the characteristic grain size by entering a value between 0.0 and 1.0. Enter 0.85 in this example. This allows variation from one particle cluster to the next. Finally, the last column specifies the material density (enter 2,600 kg/m³). This entire line of data defines the start time and conditions for the source.

Select the entire line just created in the *Line Source Property* dialog box. Copy and paste the data to the second line. Change the time of the second line to 11 February 2004 as shown in Figure 5. Note slashes are used in the dialogs for specifying time stamps (i.e., 2/11/2004 12:00:50 AM in place of 11 February 2004). This defines the end time for this source to be 50 sec after it was turned on. By this time, approximately 500 particles (50 sec * 10.0 particles/sec) have been generated. Particle generation can also be designated as time varying by changing the input parameters (such as the rate) in the second line.

Copy the entire line to the third line. Change the time of the third line to 2/11/2004 12:00:51 AM, and change the rate to 0. This turns off the source. Click on the OK button to complete the specification of particle sources. In this example, the first line starts a source, the second line specifies the duration of the source to remain active, and the third line shuts down the source. It is recommended that users follow this standard procedure for specifying sources to the PTM.

SET MODEL PARAMETERS: The user must also specify the model parameters for a PTM simulation. Follow these steps:

- Select the *Particle* module .
- Select the *PTM/New Simulation* command. This will create an empty simulation that shows up in the *Project Explorer* with the name “PartSet.”
- Right-click on the new simulation in the *Project Explorer* and select *Rename*. Change the name of the particle set to “Tides.”
- Select the *PTM/Model Control* command. This displays the *PTM Model Control* dialog (Figure 6) with the *Hydrodynamic, Sediment, and Source inputs* tab active. Parameters and file names must be provided in several of these tabs for a PTM simulation. These are described below.

In the *Hydrodynamic, Sediment, and Source inputs* tab, specify the existing files that will be used as input for the PTM simulation. Click on the file icon next to *Geometry* and select “estuary.grd.” This prompts the PTM to use this grid file to define its Eulerian domain. Click the icon next to *XMDF flow file* and select the file “estuary_tide.h5.” This prompts PTM to use this file as a source of hydrodynamic data. In the *WSE* combo box, choose the data set “tide only wse” and in the *Velocity* combo box choose the “tide only velocity.”

At the bottom section of the dialog, select the “Placement Option 1” coverage (just created) as the PTM *Coverage*. This coverage includes the line sources defined above. The SMS will convert the sources defined in that coverage to an input source file for PTM. It will have the same name as the PTM simulation with the extension “.source.”

- In the *Hydro, Sediment, and Source inputs* tab, click on the *Create files from data* button. A dialog will appear (Figure 7). This allows the user to create the input files for the PTM. The geometry and flow files already exist and do not need to be created here. However, a sediment file must be created to define the native sediments. Click on the “*Select...*” button in the top section and select the ADCIRC mesh that is loaded into SMS. This assures that this grid is the computational domain that will be used for the PTM. Set the *Solution time format* to seconds. Because the grid is already in ADCIRC fort.14 format, and the flow data are already in an XMDF file, do not select the toggles to create geometry and flow files.

Select the *Create PTM sediment file* toggle (near the bottom in Figure 7). The SMS will create a sediments file based on user input. In this example, no spatially varied data sets exist, so choose “Constant” and “mobile bed” and enter the three values of native bed sediment parameters as shown in Figure 7. If data sets defining varied native sediment sizes had existed, the user could set the toggle box to “Varied” and choose the appropriate data sets. The grain size values are used by PTM to compute roughness and entrainment coefficients. Next, specify a name for the sediment file just generated by clicking on the file icon at the bottom of the dialog, assigning the name as “estuary” for the sediment file name (“.sediments” will be added as the extension). Click the OK button to return to the model control dialog.

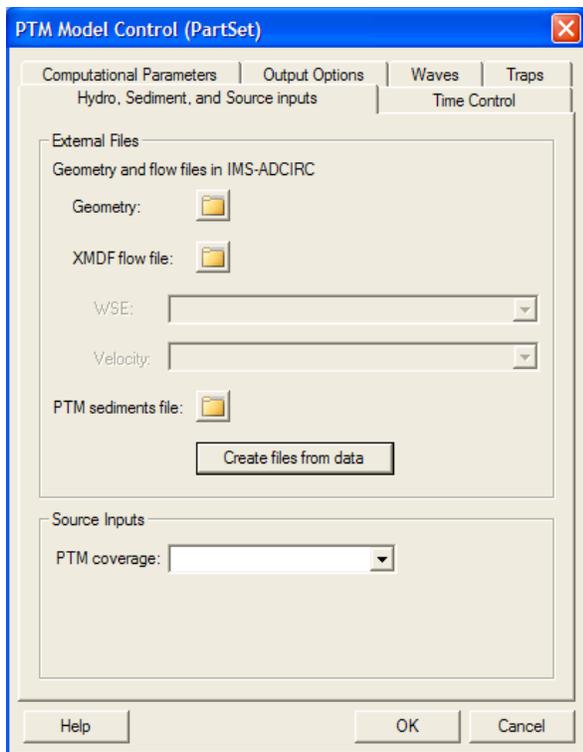


Figure 6. Selection of Hydro, Sediment, and Source inputs for the PTM

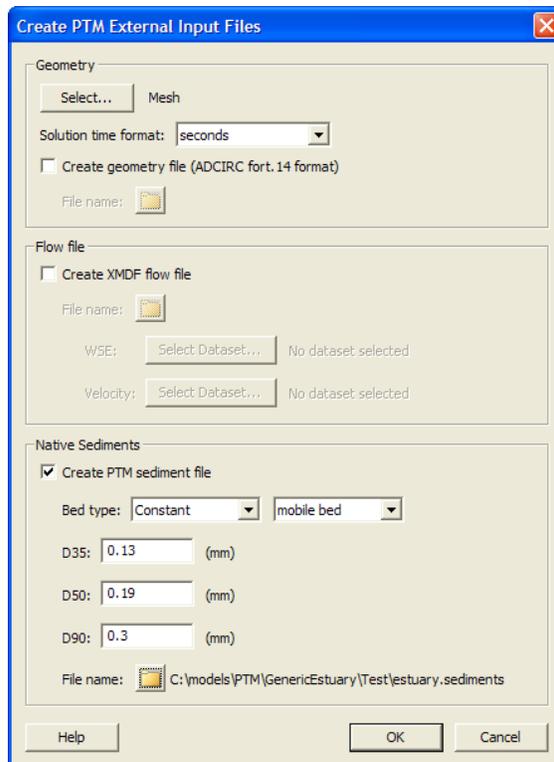


Figure 7. Defining data sources for the PTM

- Switch to the *Time Control* tab (Figure 8). This dialog allows the user to coordinate the PTM simulation with the hydrodynamics and specify a computational time-step. Enter 2/11/2004 12:00:00 AM for the *Start date/time* as the starting time for the PTM simulation. This is the earliest time any particle source is activated. Next, enter 2/12/2004 07:45:00 AM for the *End date/time*. This is the time that the PTM simulation will end. The PTM will simulate almost 32 hr of particle motion. Set the computation time-step to 3 sec. This tells the PTM to compute particle locations every 3 sec. Enter 2/09/2004 12:00:00 AM for the *Start date/time of the hydro data*. This is the reference time for the ADCIRC simulation.
- Switch to the *Computational Parameters* tab (Figure 9). Change the *Distribution* to *By weight*, and *Advection* to *Q3D* (quasi-3D) (see Demirbilek et al. (2005a) for more information on advection modes). Set the *Update shears, bedforms and mobility every* field to 300 time-steps. This prompts PTM to update the bed every 15 min. More frequent updating can slow down computation. The other values can be left as the defaults, but users can change these as necessary.

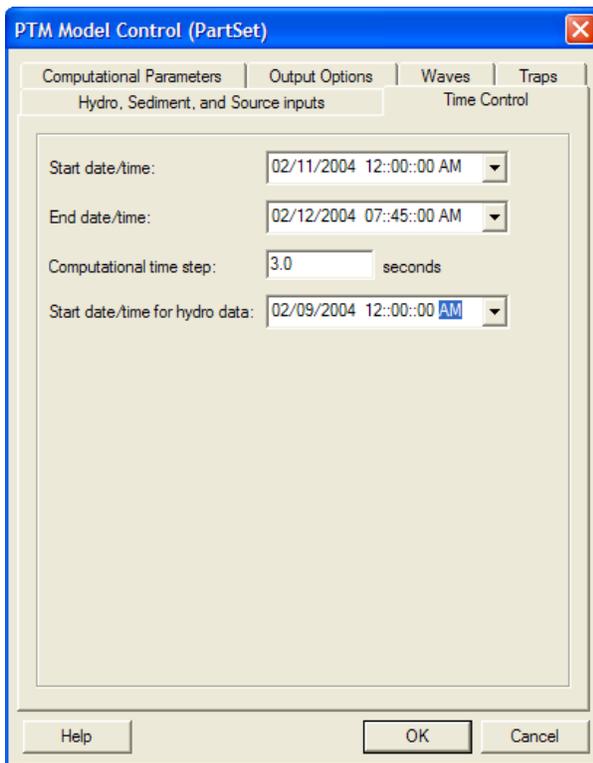


Figure 8. The PTM Time Control Dialog

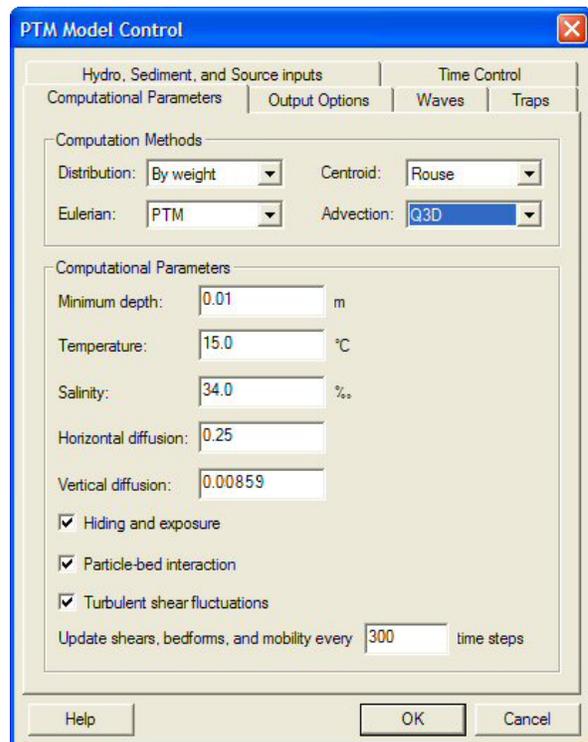


Figure 9. The PTM Computational Parameters Dialog

- Finally, switch to the *Output Options* tab (Figure 10) and set the *Grain Size*, *Mobility*, *State*, and *Parcel Mass* toggles to “on.” This prompts the PTM to output these attributes for the particles that are being traced. The SMS can display PTM output according to any of these settings after the simulation is complete. For example, the particles could be color-coded based on their representative grain size if the *Grain Size* toggle is on. Also enter 10 for the *Output Increment* to save particle locations at every 10 time-steps or 30 sec. This completes the setup of the model parameters for the simulation. Click the *OK* button. The *Traps* and *Waves* tabs will not be used in this example. In an example in the next section, waves will be

included. The PTM user guides (Demirbilek et al. 2005a, 2005b) provide more details about options that are available for using these tabs.

SAVE THE SIMULATION: The simulation data specified in these dialog boxes must be saved to files before executing the PTM. If the user forgets this step, the SMS will prompt to save the simulation before allowing the simulation to run. To save a simulation, select *File/Save New Project* and enter the name “estuary” as the name of the SMS project file (.sms).

RUN THE PTM: The simulation is ready to be run. This simulation takes approximately 10 to 15 min to compute on a PC. In the first few minutes, the PTM is creating an element neighbor table. During this time, it may appear that the application has stopped, but it is still working. To launch the PTM model, select the *PTM/Run Model* command. If

needed, the SMS will bring up a prompt to allow the user to select the directory location of the PTM executable file. Once the model location is defined, select the *OK* button to launch the model. The model location may be set in the preferences, in which case the SMS will launch the model directly after the command is given. A dialog appears that displays a progress bar of the model execution status. When the run is complete, the button at the bottom of the dialog changes from “Abort” to “Exit” and the words “Model Finished” will appear in the output window. Select the *Exit* button to close the model execution.

READ THE SOLUTION: The PTM creates two solution files. The first file stores the particle data (Lagrangian), and the other stores the mapped Eulerian data. These are named by adding file descriptors to the simulation name. For this case, the file “estuary_Tides_particles.h5” includes the particle paths and attributes. If any options for mapping solution data to the Eulerian grid had been selected, the PTM would have written a file named “estuary_Tides_maps.h5.” Both of these files are created in XMDF format (Jones et al. 2004). When the model simulation is completed, the SMS prompts the user to read in the solution files. The solution may also be read in by selecting the *File/Open* command from the menu bar, and by choosing the binary solution file “estuary_Tides_particles.h5.” As the SMS loads this file, data sets appear in the *Project Explorer* (data tree). The SMS will re-draw (frame) the active screen, since new data have been added to the project.

VISUALIZE THE PARTICLES: The SMS supports multiple methods of displaying the particles as demonstrated in this section. To view the options, select the *Display/Display Options* command (Figure 11). The left side of the display options dialog allows the user to

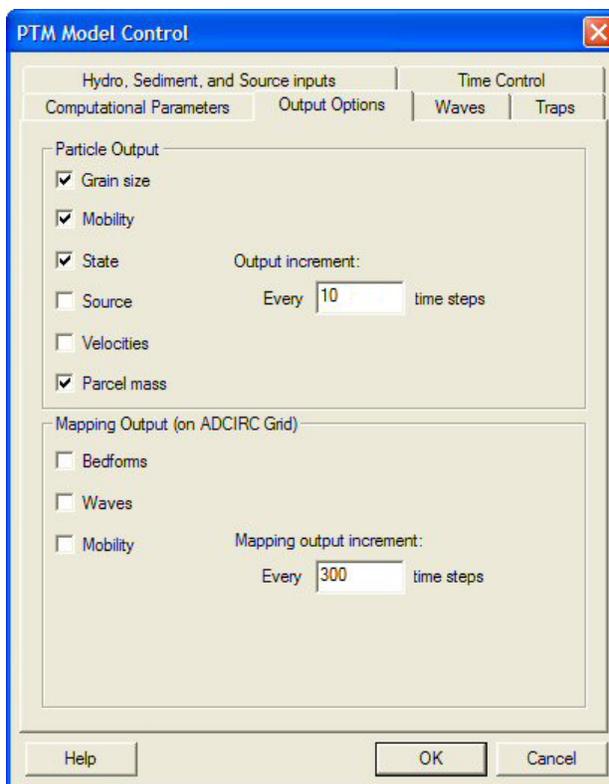


Figure 10. The PTM Output Options Dialog

control the display of the particle head and tail. Turn both of these on. Click on the symbol next to the *Particles* toggle and change the symbol to a 4-pixel diamond (as shown). The particles and their tails can be color coded. This coding can be based on any of the particle attributes or data sets saved by PTM. For example, the user may want to have the particles colored based on the characteristic grain size of that particle.

If there are multiple sources, the user may want to display particles generated at each source in a different color.

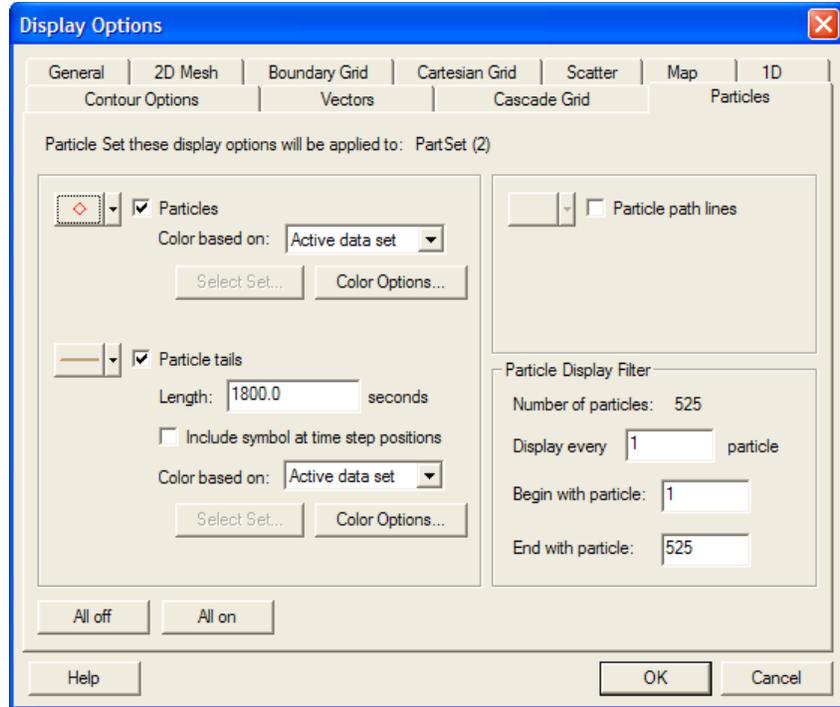


Figure 11. Particle Display Options Dialog

Each of these attributes or data sets is displayed in the project explorer. The “active” data set or attribute is displayed in bold text. For this example, change both the *Particles* and the *Particle tails* to display the “Active data set.” This tells the SMS to color the particles based on whatever attribute/data set is selected. Set the *Particle Tail Length* to 1,800 sec (30 min). This option allows the user to control the length of the particle tail. The tail indicates the speed of the particle. In the upper right corner of the dialog, turn off the toggle to display *Particle pathlines*. This option displays the entire path in a user-selected color. Normally, this option has the tendency to clutter the display, but it is useful in indicating general flow patterns.

The options in the lower right corner of the dialog allow the user to reduce the number of particles displayed. For this case, the default of viewing all the particles is appropriate. Click the *OK* button to exit this dialog.

Stepping Through Time. Select the “grain size” data set in the project explorer. Particles will be displayed in red (smaller characteristic grain size) to yellow, green, and blue (increasingly higher grain size). Click the time-step for 2/11/2004 08:00:00 AM to set the time to about 8 hr into the simulation. Use the down arrow to step through the time-steps and watch the particles move. Note that updating the time in the particle module does not currently update the time in the mesh module, so the vectors will not change. The feature to update the vector field will be added with a future release. The vectors do change when generating an animation as will be shown below. Figure 12 shows the particles approximately 8 hr into simulation time.

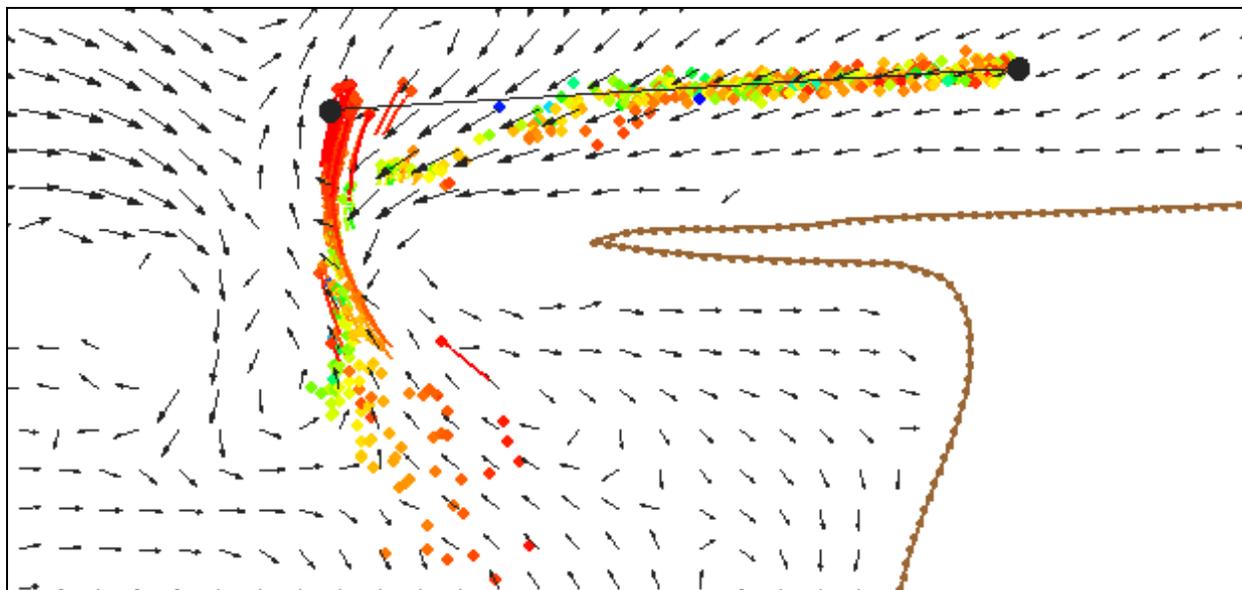


Figure 12. State of particles after 8 hr

Creating an Animation. The SMS includes the tools required to create animations of the particles and flows. When creating an animation, the SMS loops through a time range to create a user-controlled number of snapshots or frames, and saves these images to an “avi” file. The animation sequences are illustrated next:

- Ensure you are viewing the region of the domain you wish to see. Select a region similar to that shown in Figure 12 to see the range of particle motion.
- Select the *Data/Film loop.* command. The dialog that appears allows you to select the type of film loop and the file name to save. Use the defaults and click on the *Next* button.
- The start time and end time should be set to match the entire simulation. To tell the SMS to animate the Eulerian data, be sure to select the *Animate Mesh Data* toggle and enter the reference hydro time as 02/09/2004 12:00:00 AM (the starting hydro time). Select the option to *Specify Number of Frames* and enter 100. Click *Next*.
- Click on the *Clock Options* button. In the dialog that appears, select *Show Progress Bar* and choose the type to be “Horizontal Progress Type.” Set the size to 50 and click *OK*. Click *Finish* in the *Film Loop Setup*.

As the animation is generated, which may take a few minutes, it is also saved to the disk. After the generation is completed, the SMS launches the animation in the viewer PAVIA.exe that is distributed with the SMS. The animation shows that particles in the mouth of the estuary are washed into and then back out of the estuary and distribution of particles varies temporally and spatially. As the velocities decrease in the mouth of the estuary, smaller particles also start settling down. Users should close the animation after viewing and continue with the tutorial.

WAVES: The existing PTM simulation will now be copied and modified to include waves. To include waves, users must have results from a previously run STWAVE simulation, and possibly

a modified hydrodynamic output that included the wave data. The PTM will then be rerun using both the modified hydrodynamic solution and the wave solution data. The steps to be followed are described below:

- Open the STWAVE simulation by selecting the *File/Open* command and select *estuary_stwave.sim* (also included with SMS version 9.0 tutorials). Click the *OK* button. This provides the basis for wave data that will be included in the PTM simulation. You may need to go into the display options and turn off the cells of the stwave grid.
- Open the ADCIRC solution that was run to include the influence of waves by selecting the *File/Open* command. Select *estuary_waves.h5* (also included with SMS version 9.0 tutorials). Click the *OK* button.
- Right-click on the PTM simulation in the *Project Explorer* (tree) and select “Copy Simulation.” This will create a copy of the simulation just constructed.
- Right-click on this new simulation, which will be named “PartSet (2)” automatically by the SMS, and select the Rename command. Change the name to “Waves.”
- Select this new simulation, by clicking on its name in the *Project Explorer*. Now choose the *PTM/Model Control* menu command to modify simulation input.
- In the *Hydrodynamic, Sediment, and Source inputs* tab (Figure 6), select the ADCIRC solution sets that include the influence of waves on hydrodynamics. Next to the XMDF flow file, click on the icon and choose the file *estuary_waves.h5*. In the WSE combo box, select “waves wse” for water surface elevation. Similarly, select “Datasets/waves velocity” in the *Velocity* combo box. The sediment sources and grid files can remain the same.
- Click on the *Waves* tab (Figure 13). Check the toggle box to use waves, and click on the *From STWAVE Grid* button. This fills in the origin and orientation of the wave simulation based on the *STWAVE* simulation, which was loaded into the SMS in step 1. Click on the icon in the *Solution data* section and select the file “*estuary_waves.wav*.”
- In the lower portion of the dialog, enter the *Start Date/Time* of 2/09/2004 12:00:00 AM. This specifies that the wave data shall start at the same time as the hydrodynamic data.
- Enter the *Time between spectra* as 02:00:00 (i.e., waves are updated every 2 hr) and the *Number of Spectra in File* as 50 (i.e., 50 wave conditions are simulated). Note that this

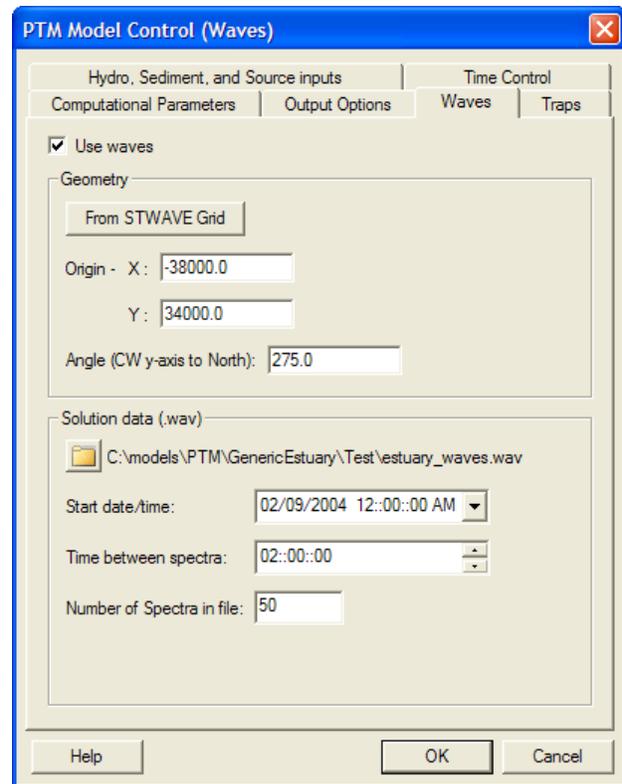


Figure 13. Wave specification in the PTM

information comes from the STWAVE run. Click the *OK* button to exit the *Model Control* dialog.

SAVE THE SIMULATION: Now resave the simulation before running the PTM. Select *File/Save* to update the estuary.sms project file saved earlier.

RUN THE PTM: The simulation with waves is ready to be run. This simulation takes approximately 15 min to compute on a 3-GHz PC. Read in the solution of this simulation using steps described earlier and view the particle motion as affected by waves. The solution file name is “estuary_Waves_particles.h5.” User will note that the wave action greatly influences particle activity near the placement site.

MAP THE PARTICLES TO A GRID: The results of the PTM simulation can also be transferred onto a Cartesian grid for visualization of the Eulerian output. Presently, this methodology only supports the particle counts, and will be enhanced in the future to create data sets for deposition depth, suspended sediment mass, and suspended sediment concentration. To utilize this feature, a Cartesian grid must be created or loaded into the SMS. This example will use the STWAVE grid to visualize the particle counts as follows:

- In the *Particle* module  select the *Data/Compute Grid Data Sets* command. This creates data sets for the Cartesian grid that can now be contoured.
- Switch to the *Cartesian Grid* module . In *Display Options*, turn on Cartesian grid contours. You can now select various time-steps to see the number of particles in each cell in the grid.

SUMMARY: This technical note provides instructions for developing a PTM simulation. A typical project could include many particle sources and wave and velocity fields. In such cases, it would be necessary to perform multiple runs and some variation in the procedure and tools described here would be required. The development of the PTM is ongoing, and the interface is also expected to change. Future technical notes will describe changes to the updated versions of the model and their SMS interface. Feedback and suggestions from users are welcome on the design, implementation, and usage of the PTM and this tutorial.

POINTS OF CONTACT: This technical note was written by Dr. Zeki Demirebilek, email: Zeki.Demirebilek@erdc.usace.army.mil), Tel: 601-634-2834, Fax: 601-634-4033; Mr. Jarrell Smith (Jarrell.Smith@erdc.usace.army.mil) of the U.S. Army Engineer Research and Development Center (ERDC), Coastal and Hydraulics Laboratory, and Dr. Alan Zundel (zundel@byu.edu) and Mr. Russell Jones (rjones@byu.edu) of Brigham Young University, and Dr. Neil MacDonald and Mr. Michael Davies of the Pacific International Engineering, PLLC. Questions about this technical note can be addressed to Dr. Demirebilek or to the Program Manager of the Dredging Operations and Environmental Research (DOER) Program, Dr. Robert M. Engler (601-634-3624, Robert.M.Engler@erdc.usace.army.mil). This technical note should be referenced as follows:

Demirbilek, Z., Smith, S. J., Zundel, A. K., Jones, R. D., McDonald, N. J., and Davies, M.E. (2005). "Particle Tracking Model (PTM) in the SMS: III. Tutorial with examples," *Dredging Operations and Engineering Research Technical Notes Collection* (ERDC TN-DOER-D6), U.S. Army Engineer Research and Development Center, Vicksburg, MS. An electronic copy of this TN is available at (<http://el.erd.usace.army.mil/dots/doer/>.)

REFERENCES

- Demirbilek, Z., Smith, J. S., Zundel, A. K., Jones, R. D., MacDonald, N. J., and Davies, M. E. (2005a). "Particle Tracking Model (PTM) in the SMS: I. Graphical interface," *Dredging Operations and Engineering Research Technical Notes Collection* (ERDC TN-DOER-D4), U.S. Army Engineer Research and Development Center, Vicksburg, MS.
- Demirbilek, Z., Smith, J. S., Zundel, A. K., Jones, R. D., MacDonald, N. J., and Davies, M. E. (2005b). "Particle Tracking Model (PTM) in the SMS: II. An overview of features and capabilities," *Dredging Operations and Engineering Research Technical Notes Collection* (ERDC TN-DOER-D5), U.S. Army Engineer Research and Development Center, Vicksburg, MS.
- Jones, N.L., Jones, R.D., Butler, C.D., and Wallace, R.M. (2004). "A generic format for multi-dimensional models", *Proceedings World Water and Environmental Resources Congress 2004*. (<http://www.pubs.asce.org/WWWdisplay.cgi?0410405>)
- Luetlich, R.A., Jr., Westerink, J.J., and Scheffner, N.W. (1992). "ADCIRC: An advanced three-dimensional circulation model for shelves, coasts, and estuaries," Technical Report DRP-92-6, U.S. Army Engineer Waterways Experiment Station, Coastal and Hydraulics Laboratory, Vicksburg, MS.
- MacDonald, N. J., and Davies, M.E. (2005). "Particle Tracking Model (PTM)," *Dredging Operations and Engineering Research Technical Report DOER-TRxx*, U.S. Army Engineer Research and Development Center, Vicksburg, MS.
- Smith, J. M., Sherlock, A. R., and Resio, D. T. (2001). "STWAVE: STEady-state spectral WAVE model: User's manual for STWAVE Version 3.0," Supplemental Report ERDC/CHL SR-01-1, U.S. Army Engineer Research and Development Center, Vicksburg, MS.
- Zundel, A.K. (2005). "Surface-water modeling system reference manual, version 9.0" Brigham Young University Environmental Modeling Research Laboratory, Provo, UT. (http://www.ems-i.com/SMS/SMS_Overview/sms_overview.html)
- Zundel, A. K., Fugal, A.L., Jones, N.L., and Demirbilek, Z. (1998). "Automatic definition of two-dimensional coastal finite element domains," *Proc. Hydroinformatics98* (eds. V. Babovic and L. C. Larsen.), A. A. Balkema, Rotterdam, 693.

NOTE: The contents of this technical note are not to be used for advertising, publication, or promotional purposes. Citation of trade names does not constitute an official endorsement or approval of the use of such products.