5. Advanced Features in CMS-Flow

5.1 Bottom Friction and Wall Friction

5.1.1 Bottom Friction Dataset

In SMS versions 10.1 and earlier, the bottom roughness is specified using the Manning's *n* coefficient in the *Cells* tab of the *CMS-Flow Model Control* window. In SMS 11.0, the option is provided to use a roughness height, or bottom friction coefficient in addition to the Manning's coefficient. In SMS 11.0 the bottom roughness is specified in the *Bottom Friction Dataset* section of within the *Input* tab of the *CMS-Flow Model Control* window (see Figure 5.1.1). The *Bottom Friction Dataset* is specified at every computational (ocean) cell and is required for each model simulation.

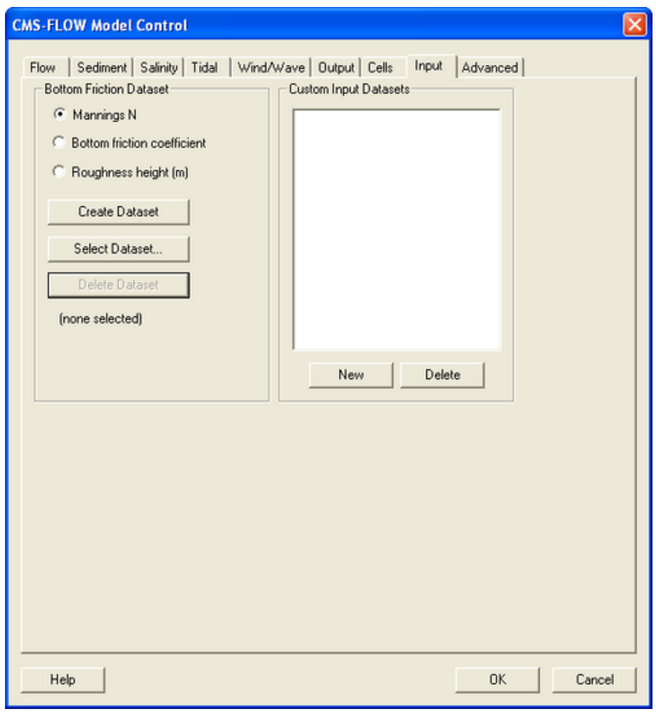

Figure 5.1.1. Specification of the *Bottom Friction Dataset* via the *Input* tab of the *CMS-Flow Model Control* window in SMS 11.0

Creating the bottom friction dataset

In SMS 11.0 each time a new CMS-Flow simulation is created a bottom friction (roughness) dataset needs to be created using the *Create Dataset* button (see Figure 5.1.2). The bottom friction parameter (related to Manning *n*) is spatially varying (cell-specific) over the grid domain. The default value upon grid creation is 0.025. At times a user may desire to represent locations where added friction is needed due to structures, vegetation, or increased turbulence due to rapid changes in current speed.

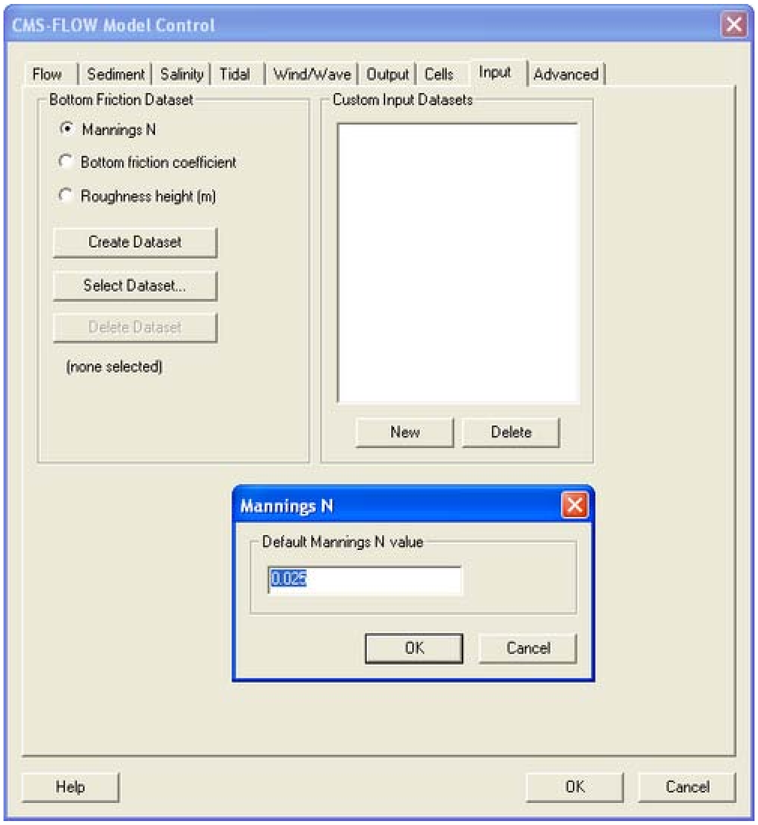

Figure 5.1.2. Initialization of the *Manning's N Dataset* in SMS 11.0

Once the *Bottom Friction Dataset* is created and the *CMS-Flow Model Control* window closed, a new auxiliary dataset will appear in the *SMS Project Explorer* (see Figure 5.1.3).

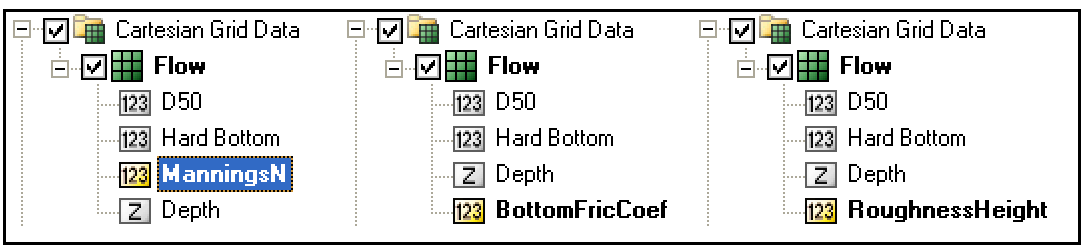

Figure 5.1.3. Three SMS Project Explorer showing the Manning's *n*, Bottom Friction Coefficient and Roughness Height datasets.

A description of the CMS-Flow cards related to the bottom friction datasets are shown in Table 5.1.1.

Table 5.1.1. SMS Cards related to the bottom friction datasets

Card	Arguments	Default	Range	Description	Versions
USE_WALL_FRICTION	CHARACTER	ON	ON | OFF	Turns on or off wall friction	>=3.5
MANNING_N_DATASET	CHARACTER CHARACTER	[<grid file>] [<grid name>//"Datasets/ManningsN"]	none	Grid file name and dataset path for the input Manning's n dataset	>=v3.5
BOTTOM_FRICTION_COEF_DATASET	CHARACTER CHARACTER	[<grid file>] [<grid name>//"Datasets/BottomFricCoef"]	none	Grid file name and dataset path for the input bottom friction coefficient dataset	>=v4.0
ROUGHNESS_HEIGHT_DATASET	CHARACTER CHARACTER	[<grid file>] [<grid name>//"Datasets/RoughnessHeight"]	none	Grid file name and dataset path for the input roughness height dataset	>=v4.0

Editing the bottom friction dataset

Bottom roughness datasets may be edited manually by selecting and modifying cells, using the *Data Calculator*, or by interpolating from a scatter set. Figure 5.1.4 shows an example where the Manning's *n* coefficient was increased for Shark River Inlet to account for the presence of vertical piles.

Figure 5.1.4. Manning's *n* contours after modifying the areas under all three bridges.

- **Interpolation of scatter to whole grid**
  1. Select the scatter set
  2. Click on the menu *Scatter* | *Interpolate to Cartesian Grid* (see Figure 5.1.5)

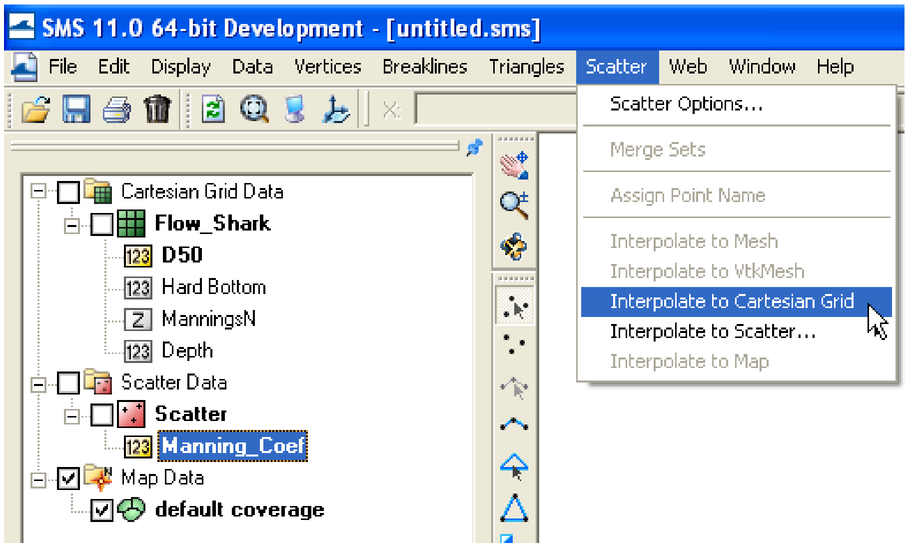

Figure 5.1.5. Interpolating a scatter set of Manning's *n* values to the CMS-Flow grid.

  3. In the *Interpolation* window, enter an extrapolation value, a name for the dataset and click *OK* (see Figure 5.1.6).

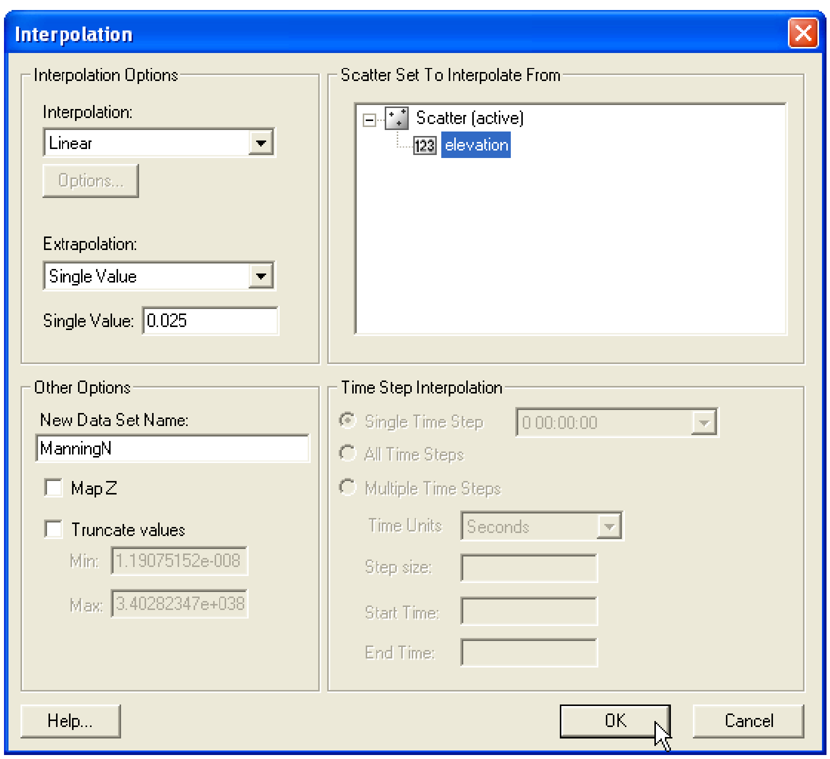

Figure 5.1.6. Interpolating a scatter set of Manning's *n* values to the CMS-Flow grid.

The dataset will be interpolated to the whole grid and grid cells outside of the scatter triangulation will be set to the extrapolation value.

In some cases, it is desirable to only interpolate a bottom roughness scatter set to a portion of the grid. However, it is only possible to interpolate portions of the grid from scatters for the *Elevation* dataset. The *Elevation* dataset is the dataset with a Z to the left. Therefore, it is necessary to *Map* the bottom friction dataset as the *Elevation* dataset, interpolate the scatter set and then re-map the actual grid elevation (usually named *Depth*) dataset as the *Elevation* dataset. To interpolate a bottom roughness dataset to a portion of a grid:

- **Interpolation to portion of grid**

    1. Click on the menu *Data* | *Map Elevation.* The bottom friction dataset will appear with the symbol Z.

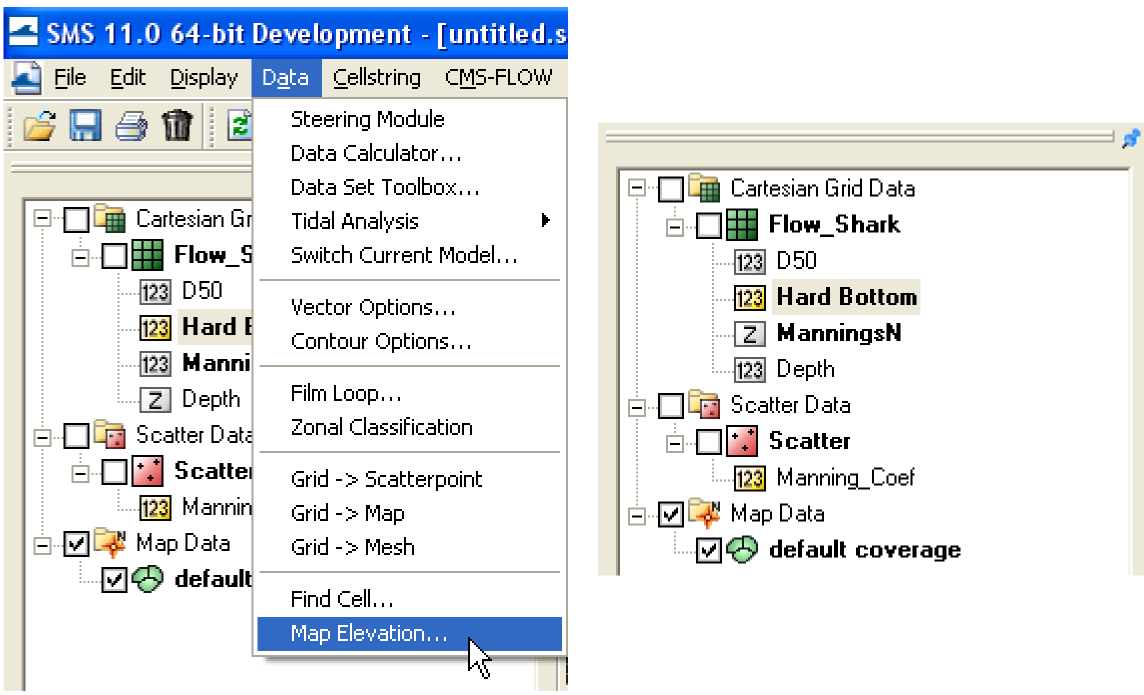

Figure 5.1.7. Mapping of the bottom friction dataset as the Elevation dataset.

2. Select the portion of the grid from which you want to interpolate the bottom friction dataset. This can be done by clicking and holding the left mouse button or by holding the Control key and single clicking so select the polygon and double clicking on the last polygon point (shown in Figure 5.1.8).
3. Once the correct cells are selected, right-click on the selected cells and select *Interpolate Bathymetry*.

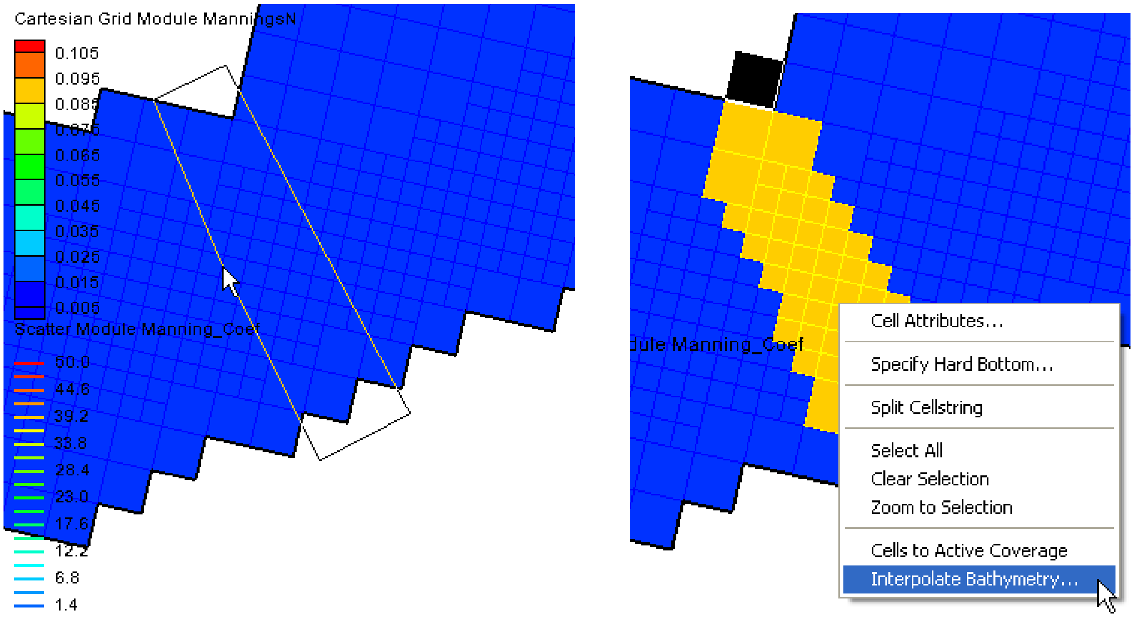

Figure 5.1.8. Mapping of the bottom friction dataset as the Elevation dataset.

4. In the interpolation window, select and appropriate extrapolation value and click *OK* (see Figure 5.1.9).

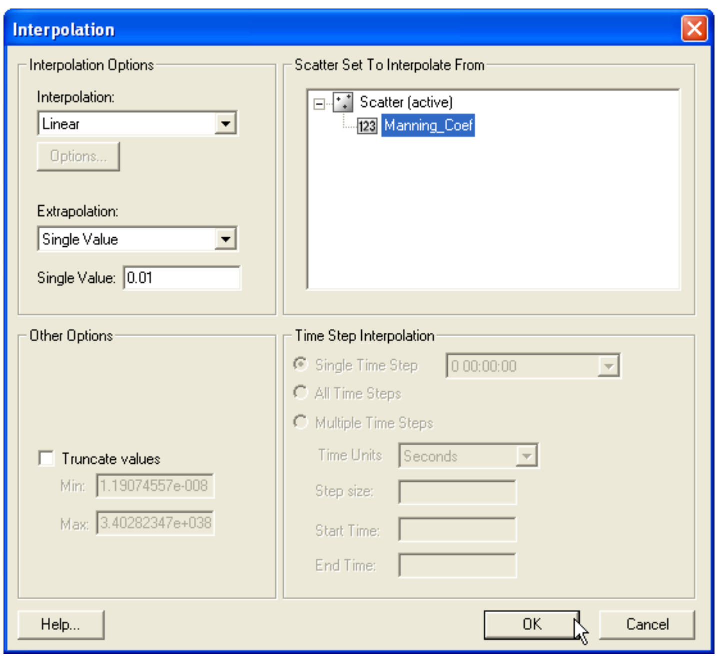

Figure 5.1.9. SMS Interpolation window.

Figure 5.1.10 shows an example of how the final bottom friction dataset looks like once a portion of the dataset has been interpolated from a scatter set.

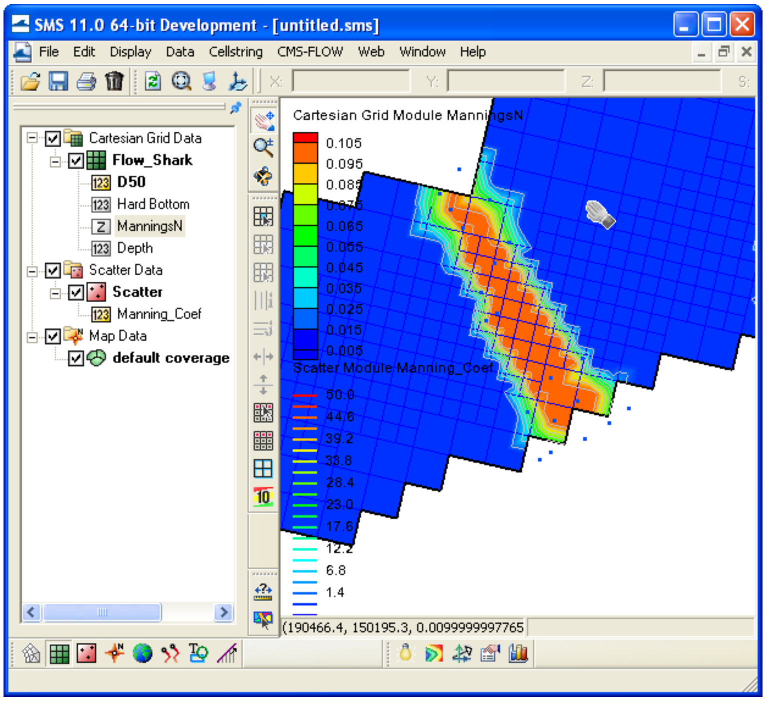

Figure 5.1.10. SMS *Interpolation* window.

5. Re-map the *Depth* dataset as the Elevation dataset by clicking on Data | Map Elevation and selecting the Depth dataset (Figure 5.1.11).

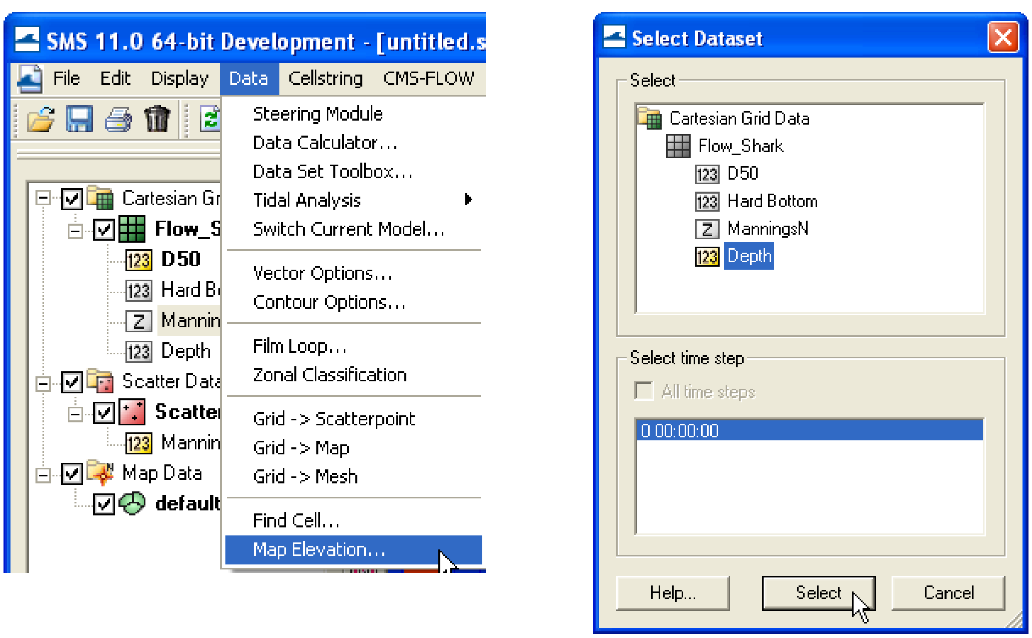

Figure 5.1.11. Mapping of *Depth* dataset as the *Elevation* dataset.

Exporting the bottom friction dataset

Exporting the bottom roughness (friction) datasets is useful for creating different project alternatives or when switching from different bottom roughness datasets types such as from *Manning's n* to *Bottom Friction Coefficient* and back. It is also useful for scripting multiple runs with different project alternatives.

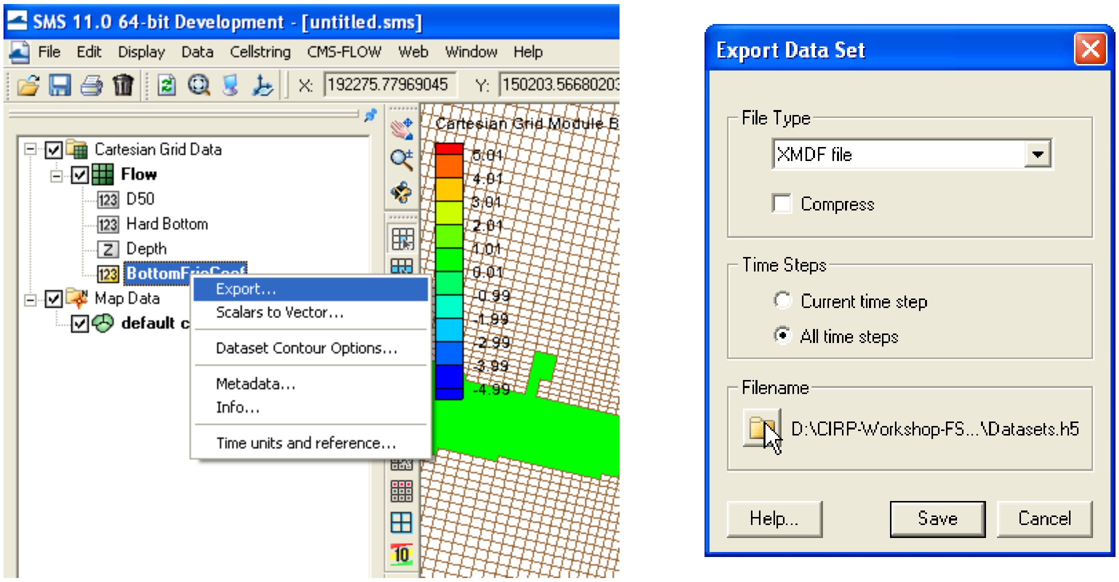

Figure 5.1.12. Manning's *n* contours after modifying the areas under all three bridges.

Additional Bottom Friction Cards

Additional advanced cards are available for setting the bottom friction to a constant for the whole grid. These cards are useful for running sensitivity analysis for a wide range of values in cases which can be approximated with a single constant bottom friction value.

Table 5.1.2. Advanced Cards to set bottom friction dataset to a constant value

Card	Arguments	Default	Range	Description	CMS Versions
MANNING_N_CONSTANT	real number	none	>=0.0	Specifies a constant input *Manning's n* coefficient. Over-rides any previous bottom friction cards.	>=v4.0
BOTTOM_FRICTION_COEF_CONSTANT	real number	none	>=0.0	Specifies a constant input *Bottom Friction Coefficient*. Over-rides any previous bottom friction cards.	>=v4.0
ROUGHNESS_HEIGHT_CONSTANT	real number	none	>=0.0	Specifies a constant *Roughness Height* in m. Over-rides any previous bottom friction cards.	>=v4.0

5.1.2 Wall Friction

The wall friction enhances the flow drag perpendicular to any dry boundary. The wall friction may be turned *On* or *Off* in the *Flow* tab of the *CMS-Flow Model Control* window (see Figure 5.1.13). The default in SMS is for the wall friction to be *ON*. Table 5.1.3 shows the CMS card used activating the wall friction.

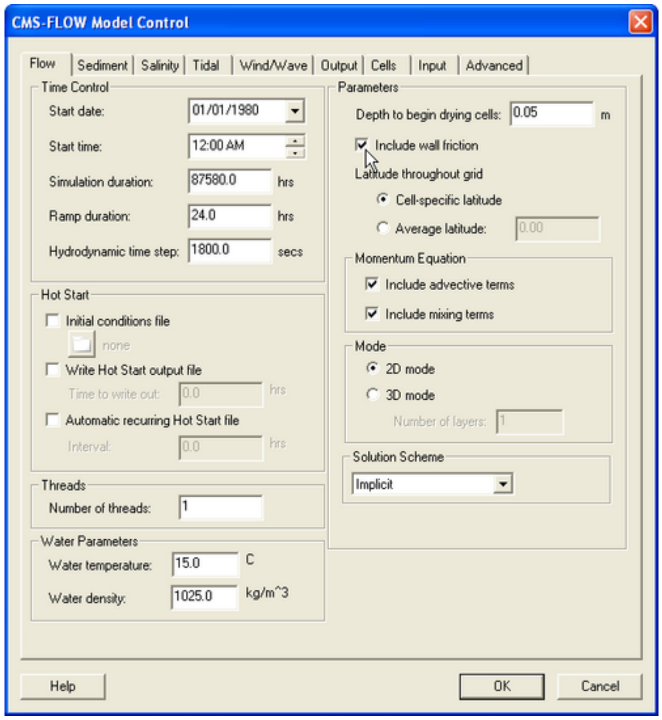

Figure 5.1.13. Wall friction specification in the SMS 11.0 *CMS-Flow Model Control* window.

Table 5.1.3. SMS Cards related to the bottom friction datasets

Card	Arguments	Default	Range	Description	Versions
USE_WALL_FRICTION	CHARACTER	ON	ON | OFF	Turns on or off wall friction	>=3.5

5.1.3 Wave-enhanced Bottom Friction (Advanced)

In the presence of waves, the mean (short-wave averaged) bottom shear stress which is used in the momentum equations is enhanced. There are 5 different options in CMS for estimating the wave-enhanced bottom friction. The default is a simple quadratic formula but can be changes using the *Advanced Card* described in Table 5.1.4.

Table 5.1.4. Advanced Card used for specifying the wave-current mean stress formulation

Card	Arguments	Default	Range	Description	CMS Versions
WAVE-CURRENT_MEAN_STRESS	CHARACTER	W09	W09 | DATA2 | DATA13 | F85 | HT95	Defines the model used for calculating the mean bottom shear stress used in hydro	>=4.0
WAVE_BOTTOM_FRICTION_COEFFICIENT	REAL	0.5	0.3-0.7	Wave bottom friction coefficient used for quadratic combined wave-current mean bed shear stress calculation.	>=4.0

5.1.4 Bed Slope Friction Factor (Advanced)

The bed slope friction factor accounts for the increased surface area over sloping beds as compared to the horizontal area. This feature may be turned *On* or *Off* through the Advanced Card described in Table 5.1.5.

Table 5.1.5. Advanced Card used for activating the bed slope friction factor

Card	Arguments	Default	Range	Description	CMS Versions
BED_SLOPE_FRICTION_FACTOR	CHARACTER	OFF	ON| OFF	Specifies whether to include the bed slope friction factor or not in the calculation of the bed friction	>=4.0

5.1.5 Example

A total of six cases are set up for Shark River inlet with combinations of Manning's *n* coefficients and wave-enhanced bottom friction formulations in order to test the model sensitivity. Three different values of Manning's coefficient were used: 0.01, 0. 02, and 0.025. The wave-current bottom friction formulation was set to either the simplified quadratic formulation with a wave coefficient equal to 0.5 (W09) or Soulsby's 2 parameter empirical relation (DATA2). The longshore current at an observation point to the north of Shark River inlet is plotted in Figure 5.1.14 for short wave event. The simplified quadratic formulation tended to produce smaller longshore velocities than Soulsby's formula and the results differed most for larger values of Manning's *n* coefficient. However, for low values of Manning's *n*, the two formulations produced similar results.

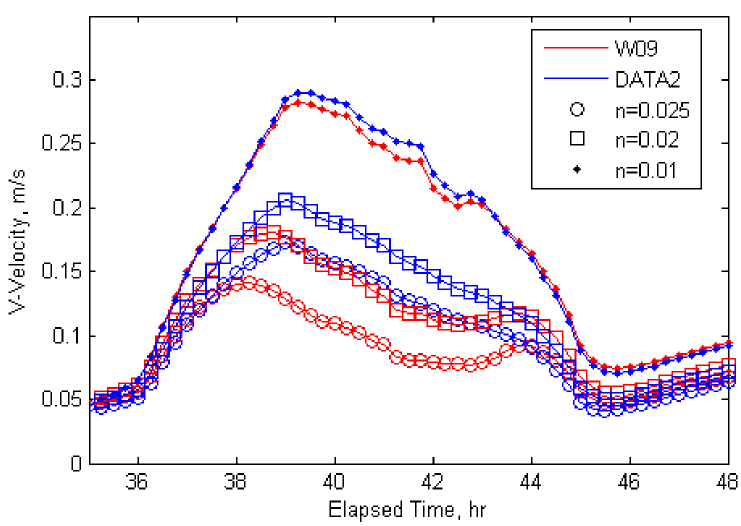

Figure 5.1.14. Wall friction specification in the SMS 11.0 *CMS-Flow Model Control* window.

5.2 Eddy Viscosity

5.2.1 Mixing Terms

Turbulent diffusion is calculated in CMS using the eddy viscosity concept. The eddy viscosity is not a user definable parameter. The diffusion or *Mixing* terms of the momentum and transport equations may be turned off in the *Flow* tab of the *CMS-Flow Model Control* window (Figure 5.2.1). The CMS card used to turn *On* or *Off* the mixing terms is described in the Table below.

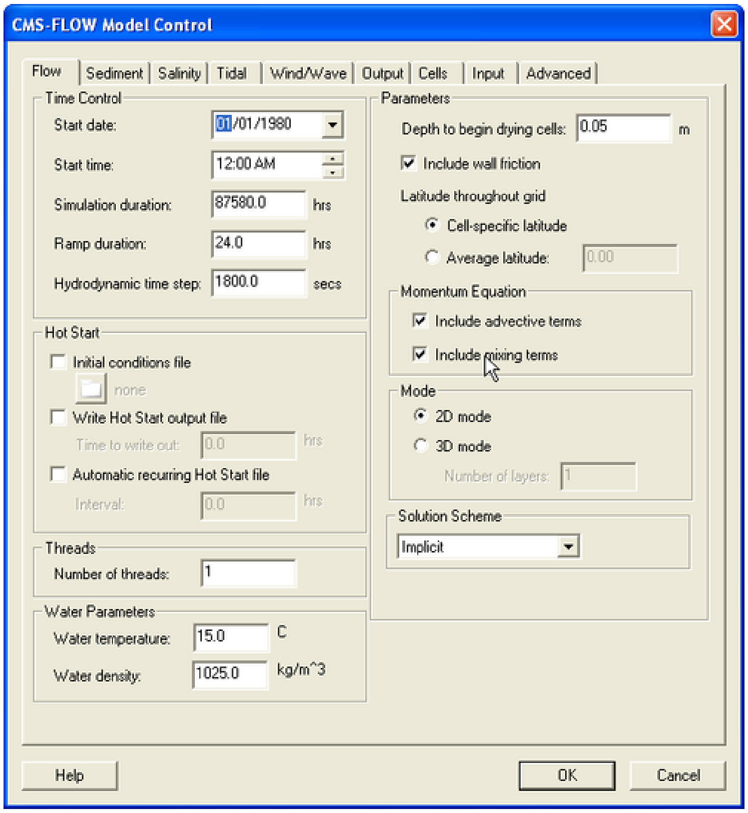

Figure 5.2.1. Toggle for turning *On* and *Off* the mixing terms in the SMS 11.0 interface.

The CMS-Flow cards used for mixing terms is described, below, in Table 5.2.1.

Table 5.2.1. CMS card used to turn *On* or *Off* the mixing terms in the momentum and transport equations

Card	Arguments	Default	Typical Range	Description	Versions
USE_MIXING_TERMS	CHARACTER	ON	ON |OFF	Turns On and Off the mixing terms in the momentum and transport equations.	>=3.5

5.2.2 Turbulence Models (Advanced)

Due to the large amount of turbulence in the surf zone, the nearshore eddy viscosity and resulting mixing terms can be quite large making the flow sensitive to the eddy viscosity formulation implemented.

A description of the advanced cards related to eddy viscosity is shown, below, in Table 5.2.2.

Table 5.2.2. CMS card used to set turbulence model and eddy viscosity terms

<table>
<tr><th>Card</th><th>Arguments</th><th>Default</th><th>Typical Range</th><th>Description</th><th>Versions</th></tr>
<tr><td>TURBULENCE_MODEL</td><td>CHARACTER</td><td>SUBGRID</td><td>SUBGRID |FALCONER| PARABOLIC | MIXING-LENGTH</td><td>Specifies the turbulence model used</td><td>>=3.5</td></tr>
<tr><td>EDDY_VISCOSITY_CONSTANT</td><td>REAL</td><td>1.0E-6</td><td>>=0.0</td><td>Constant contribution or base value of eddy viscosity</td><td>>=3.5</td></tr>
<tr><td>EDDY_VISCOSITY_BOTTOM</td><td>REAL</td><td>0.0667</td><td>0.02-0.1</td><td>Coefficient related to the contribution to eddy viscosity from the bottom shear</td><td>>=3.5</td></tr>
<tr><td>EDDY_VISCOSITY_HORIZONTAL</td><td>REAL</td><td>0.4</td><td>0.1-0.2</td><td>Coefficient related to the contribution to eddy viscosity from horizontal velocity gradients</td><td>>=3.5</td></tr>
<tr><td>EDDY_VISCOSITY_WAVE</td><td>REAL</td><td>0.5</td><td>0.2-1.0</td><td>Coefficient related to the wave bottom friction contribution to eddy viscosity</td><td>>=4.0</td></tr>
<tr><td>EDDY_VISCOSITY_BREAKING</td><td>REAL</td><td>0.05</td><td>0.05-0.1</td><td>Coefficient related to the wave breaking contribution to eddy viscosity</td><td>>=4.0</td></tr>
</table>

Notes:

1. The eddy viscosity can be calibrated by looking at several features of the flow. Increasing the eddy viscosity

    1. Increases the width of the longshore current.
    2. Increases the spread of the ebb and flood tidal jet currents.
    3. Decreases the offshore extent of the ebb jet current.
    4. Smoothes out small gyres and recirculation zones.

2. The eddy viscosity also affects the sediment mixing since the sediment mixing coefficient is calculated as the eddy viscosity divided by the Schmidt number.

5.3 Spatially Variable Wind and Atmospheric Pressure

CMS-Flow V4.0 and higher have the option to use spatially variable wind and atmospheric pressure forcing. Currently, this feature is specified in the advanced card section. A description of the cards related to winds and atmospheric pressure are described, below, in Table 5.3.1.

Table 5.3.1. CMS-Flow cards related to winds and atmospheric pressure

Card	Arguments	Type
WIND_OUT_TIMES_LIST	Id number of output times list	integer
WIND_DRAG_COEFFICIENT	kappa in Hsu (1988) (default 0.4)	real
WIND_INPUT_CURVE	Name of model parameter file followed wind curve path	character
ANEMOMETER_HEIGHT	Height of wind speeds	meter
OCEANWEATHER_WIND_FILE	Name of Oceanweather, Inc. wind file (*.win)	character
OCEANWEATHER_PRES_FILE	Name of Oceanweather, Inc. pressure file (*.pre)	character
OCEANWEATHER_XY_FILE	Name of Oceanweather, Inc. coordinate file (*.xy)	character
WIND_PRESSURE_SINGLE_FILE	Name of fleet wind and pressure file	character
WIND_PRESSURE_GRID_PARAM	Nwlat,Nwlon,wLatMax,wLonMin,wLatInc,wLonInc	integer and real
WIND_PRESSURE_TIME_INCREMENT	wTimeInc	real

- **Notes:**

  Interpolation File: When using spatially variable winds CMS V4.0 will output a file named *Intpcoef_wndfl.bin*. This file contains the coefficients used to interpolate the winds and atmospheric pressure to the CMS-Flow grid. If the same flow and wind grids are run, the model will automatically detect the interpolation file and read in the coefficients instead of calculating them. If the either grid is changed in size, the model will automatically detect this and recalculate the coefficients. However, if the grids are changed but stay the same size, the model will not be able to detect that they have changed. Therefore, whenever any changes are made to the flow or wind grids, it is best to delete the interpolation file.

  Ramp Period: During the ramp period, the ramp is applied to both the wind shear stresses and spatial variations of the atmospheric pressure.

5.3.1 Single ASCII File Format

This file format is equivalent to the ADCIRC fort.22 format with NWS=6. The winds and atmospheric pressure are specified on a rectangular grid in the same coordinate system and projection as the CMS. The grid is assumed to vary from north to south and west to east so that north-west corner is the grid origin. The spatial extents of the wind/pressure grid must be consistent with the flow grid.

5.3.2 Oceanweather Wind File

The file format for the Oceanweather wind file (*.win) is best described with an example:

```
Oceanweather WIN/PRE Format                            2008090912     2008091406
iLat= 118iLong= 201DX=0.0150DY=0.0150SWLat=28.50000SWLon=-96.0000DT=200809091200
-4.8891   -4.8873   -4.8855   -4.8838   -4.8821   -4.8805   -4.8789   -4.8774
-4.8759   -4.8744   -4.8730   -4.8717   -4.8704   -4.8692   -4.8680   -4.8669
...
-2.1530   -2.1382   -2.1233   -2.1085   -2.0937   -2.0788   -2.0637   -2.0486
-2.0335   -2.0184   -2.0035   -1.9887   -1.9741   -1.9598   -1.9458   -1.9322
-1.9190   -1.9062   -1.8938   -1.8820   -1.8707   -1.8600
iLat= 118iLong= 201DX=0.0150DY=0.0150SWLat=28.50000SWLon=-96.0000DT=200809091215
-4.9107   -4.9090   -4.9074   -4.9058   -4.9043   -4.9028   -4.9014   -4.9001
-4.8988   -4.8975   -4.8963   -4.8952   -4.8942   -4.8932   -4.8923   -4.8914
-4.8906   -4.8898   -4.8889   -4.8881   -4.8873   -4.8866   -4.8860   -4.8854
...
```

The last two arguments of the header are the starting and end times in yyyymmddhh. The wind data are preceded by a header line with the grid information. iLat and iLong are latitude North and longitude West. DX and DY are the grid spacing in units of degrees. SWLat and SWLon are the coordinates of the south-west grid point. DT is the actual time in yyyymmddhhMM. First the wind velocities in the x-direction are read then the velocities in the y-direction. The wind speeds are written in m/s.

FORTRAN Code

An excerpt of the FORTRAN code used to read the file is shown below:

```
open(nunit,file=windfile,status='old')
read(nunit,*)   !skip  header
11 format(t6,i4,t16,i4)
read(nunit,11) nwindi,  nwindj
backspace(nunit)
 !...
13  format(68x,I4,4(I2))
read(nunit,13,end=333,err=333)  iyear,imonth,iday,ihour,imin
!...
12  format(8f10.0)
read(nunit,12,end=333,err=333)  ((wndspdx(i,j),j=1,nwindj),i=1,nwindi)
read(nunit,12,end=333,err=333) ((wndspdy(i,j),j=1,nwindj),i=1,nwindi)
!...
333 close(nunit)
```

5.3.3 Oceanweather Pressure File

The file format for the Oceanweather pressure file (*.pre) is best described with an example:

```
 Oceanweather WIN/PRE Format                                2008090912     2008091406
iLat= 118iLong= 201DX=0.0150DY=0.0150SWLat=28.50000SWLon=-96.0000DT=200809091200
1012.8565 1012.8564 1012.8563 1012.8563 1012.8563 1012.8563 1012.8563 1012.8564
1012.8565 1012.8566 1012.8568 1012.8569 1012.8572 1012.8574 1012.8577 1012.8580
1012.8583 1012.8587 1012.8591 1012.8596 1012.8600 1012.8605 1012.8610 1012.8616
...
1014.2051 1014.2062 1014.2073 1014.2084 1014.2096 1014.2107 1014.2119 1014.2130
1014.2141 1014.2152 1014.2164 1014.2175 1014.2186 1014.2197
iLat= 118iLong= 201DX=0.0150DY=0.0150SWLat=28.50000SWLon=-96.0000DT=200809091215
1012.9089 1012.9089 1012.9089 1012.9089 1012.9090 1012.9091 1012.9092 1012.9094
1012.9095 1012.9098 1012.9100 1012.9103 1012.9106 1012.9109 1012.9113 1012.9117
1012.9120 1012.9125 1012.9130 1012.9135 1012.9140 1012.9146 1012.9152 1012.9158
...
```

The last two arguments of the header are the starting and end times in yyyymmddhh. The atmospheric pressure data (in units of mbar) is preceded by a header line with the grid information. iLat and iLong are latitude North and longitude West. DX and DY are the grid spacing in units of degrees. SWLat and SWLon are the coordinates of the south-west grid point. DT is the actual time in yyyymmddhhMM.

5.3.4 Oceanweather Coordinate File Format

The file format for the Oceanweather coordinate file (*.xy) should contain the coordinates of the Oceanweather grid in the same coordinate system as the CMS-Flow grid. The order of the points should be from the south-west corner along each row. A small fortran program provided here converts the Oceanweather grid information to an *.xy file. This can also be downloaded from the CIRP website: http://cirp.usace.army.mil/CIRPwiki/images/8/82/WINDLOC.rar

```
903301.20526 77626.22617
904769.21902 77663.98766
906237.22790 77701.93744
907705.23188 77740.07550
909173.23093 77778.40184
910641.22504 77816.91646
...
```

- **Notes:**

  1. The first column is the horizontal coordinate, and the second is the vertical coordinate.

5.4 Boundary Conditions

CMS-Flow has five types of boundary conditions which are listed and discussed below. The figure below shows the CMS-Flow Boundary Conditions window in SMS. All CMS-Flow boundary conditions are forced at the edges of the domain by use of cellstrings defined with the Surfacewater Modeling System (SMS). Cellstrings can either be created manually or using the SMS tool called *Generate Along Boundary* which is found under the *Cellstring* menu.

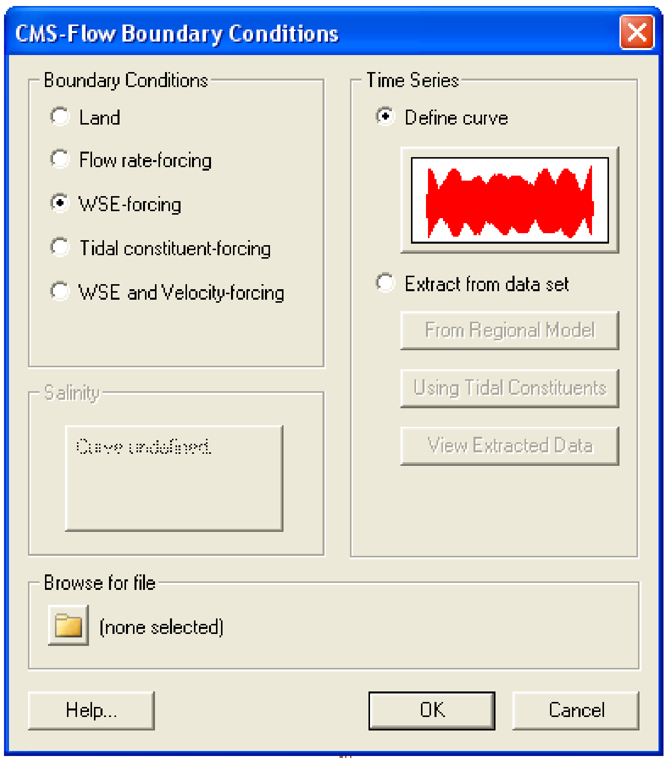

Figure 5.4.1 CMS-Flow Boundary Conditions window

Creating Cell Strings

Cell strings can either be manually created by clicking on the cells or by using the SMS interface to automatically detect and assign the cell strings.

- To manually create a cellstring

  1. Select the *Create cellstring* tool.
  2. Single-click on the first cell of the cellstring.
  3. If necessary, single click on subsequent cells in the cellstring (around corners).
  4. Double-click on the last cellstring to finish.

- To automatically generate cellstring in SMS

1. Click on *Cellstring | Generate Cellstrings* and select one of the options:
    1. *Along Open Boundaries*
    2. *Along Land Boundaries*
    3. *Along Open and Land Boundaries*

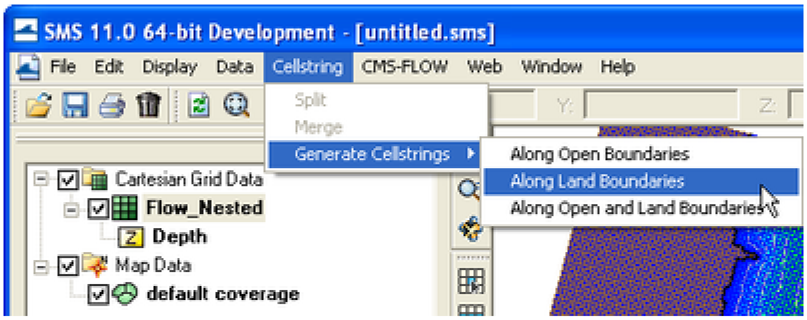

Figure 5.4.2. Automated cellstring generation in SMS 11.0

- **Notes:**

1. Cell strings must be at least three cells wide.
2. Cell strings must cover ALL open boundaries. Leaving any open boundaries without an assigned boundary will close that boundary and apply a wall boundary condition.
3. Land boundary conditions are not necessary at close boundaries (cells next to inactive cells). The CMS will automatically detect these land boundaries.
4. The orientation of cellstrings is important. Some inflow and all outflow boundary conditions assume a zero-gradient in the normal direction. If this assumption is not valid, the boundary may produce instability problems. It is recommended that the boundaries be placed in a orientation such that the spatial gradients normal to the cellstring or minimal.

Land Boundary

The land boundary condition is equivalent to a zero flux boundary condition. The land boundary condition is the default boundary condition when a cell string is created. It is not necessary to define land boundaries in the interior of the CMS-Flow domain, since land boundaries will automatically be detected by CMS-Flow based on the local water depths and the presence of inactive land cells.

Flow Rate-Forcing

The flow rate boundary condition specifies a time series of water fluxes in units of m^3/s per cell.

Notes:

- Total flow rate specified is divided between the total number of cells in the cellstring with each carrying a portion of the total.
- This boundary type may only be specified along cell strings which are straight.
- The sign of the flow rate curve is dependent on the direction of flow with respect to the origin (always lower-left hand corner of the grid).

  This guide should assist in proper assignment:

    - Flow rate from the East - Negative value
    - Flow rate from the West - Positive value
    - Flow rate from the North - Negative value
    - Flow rate from the South - Positive value

Water Surface Elevation Forcing

Two types of Water Surface Elevation Forcing exist for CMS-Flow. Once a water surface elevation curve (or series of curves) is applied, the user is able to display the curve information graphically.

Single time series curve for each cell string (Option "Define curve" in SMS)

For this type of BC, a single time series of water levels is specified and is applied at all of the cells along a boundary cell string. The time series curve may be specified by either importing an SMS *.xys file, copying tabular data into SMS, or manually entered the time series information in SMS. To assign this type of BC in SMS:

1. Click on the *Select a Cellstring* button, right-click the blue cellstring along the ocean boundary (if this cannot be selected it is because all cellstrings were deleted; this can be regenerated by clicking and clicking around the ocean cells or selecting *Cellstring* and *Generate Along Boundary*), select *Assign BC*.
2. Select the *WSE-forcing* boundary condition, and click curve undefined.
3. Copy-paste a time series into the *XY Series Editor*.
4. Click on the *Export* button to save the water level time series to an SMS xys file (time series file).
5. Save the *SMS project File* (*.sms) or *CMS-Flow Simulation File* (*.cmcards).

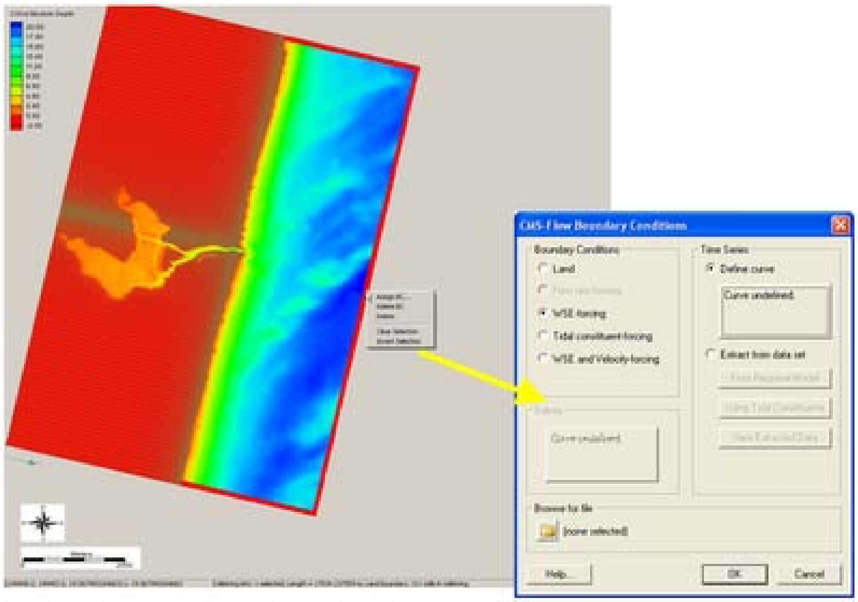

Figure 5.4.3. Selecting the cellstring along the ocean boundary, and clicking OK

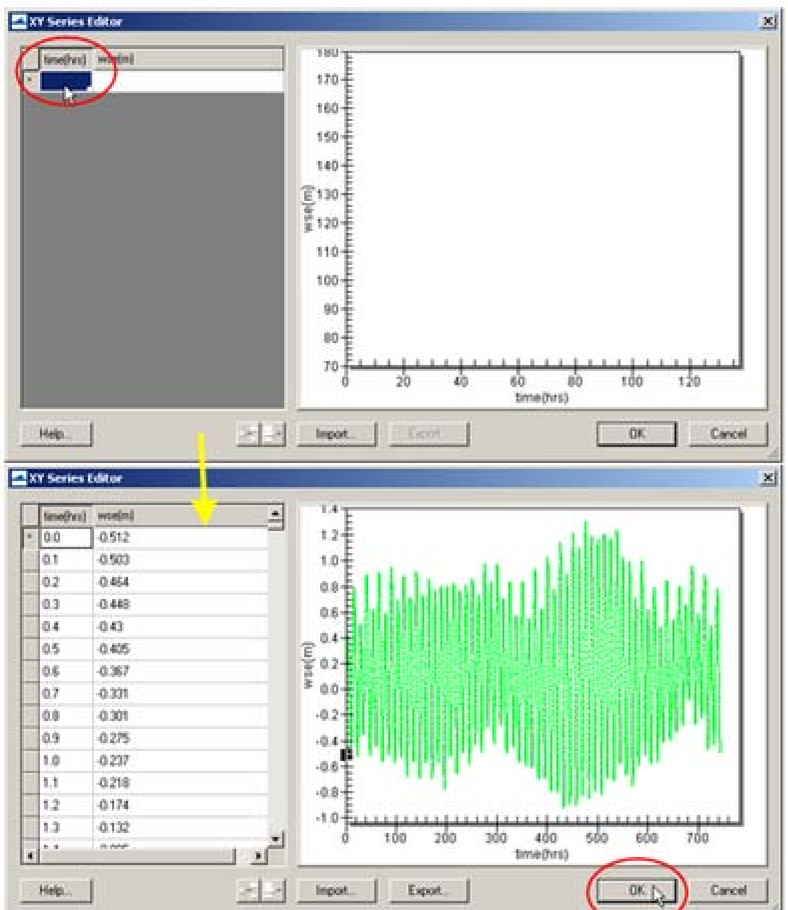

Figure 5.2.4. Top: is the blank '''*XY Series Editor'''*; Bottom: after the data are pasted

Time series curve for each cell on cell string (Option "Extract from data set" in SMS)

User creates a cellstring for the given boundary and extracts multiple time series curves from a dataset or database. Each cell along the cellstring is given its own time-series curve information. Examples are:

- Extraction of water surface elevation values from a larger domain solution (ie. Larger CMS-Flow or ADCIRC grid)
- Extraction of tidal constituent information from a tidal database, from which a water surface elevation curve can be generated.

Tidal Constituent Forcing

CMS-Flow has the option of forcing with a harmonic tidal water surface elevation. To assign a boundary as a tidal boundary:

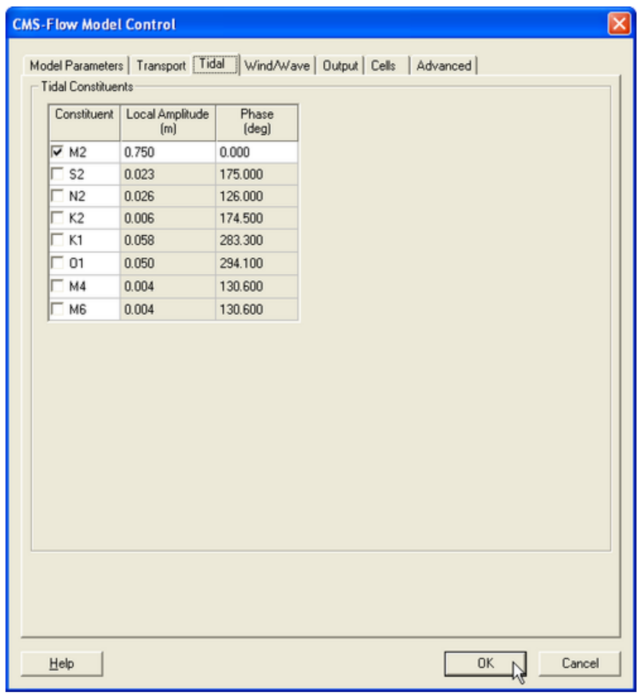

Figure 5.2.5. Tidal Constituent forcing window in SMS

1. Select a cellstring
2. Click under *CMS-Flow | Assign BC*
3. In the *CMS-Flow Boundary Conditions* window, click on *Tidal constituent-forcing*, and click *OK*.
4. Open the *CMS-Flow Model Control* window and click on the tab labeled *Tidal*.
5. Enter the amplitudes and phases of the tidal constituents. Note that the same amplitude and phase are applied to entire cellstring and click *OK*

Table 5.4.1 Tidal constituents used in CMS

Constituent	Speed	Constituent	Speed	Constituent	Speed	Constituent	Speed
SA*	0.041067	SSA*	0.082137	MM*	0.54438	MSF*	1.0159
MF*	1.098	2Q1*	12.8543	Q1*	13.3987	RHO1*	13.4715
O1*	13.943	M1*	14.4967	P1*	14.9589	S1*	15.0
K1*	15.0411	J1*	15.5854	OO1*	16.1391	2N2*	27.8954
MU2*	27.9682	N2*	28.4397	NU2*	28.5126	M2	28.9841
LDA2*	29.4556	L2*	29.5285	T2*	29.9589	S2	30
R2*	30.0411	K2	30.0821	2SM2*	31.0159	2MK3*	42.9271
M3*	43.4762	MK3*	44.0252	MN4*	57.4238	M4	57.9682
MS4*	58.9841	S4*	60.0	M6	86.9523	S6*	90.0
M8*	115.9364						

* Only available through advanced cards for CMS >v4.0

Water Surface Elevation and Velocity Forcing

When nesting a CMS-Flow grid within a larger CMS-Flow or ADCIRC grid, it is possible to extract both water levels and velocities from the larger grid. These values are internally converted into fluxes at the cell faces.

- To extract water surface elevations and velocities from a larger grid::

    1. Load both the nested CMS-Flow grid.
    2. Load the larger CMS-Flow or ADCIRC grid along with the solution file.
    3. Select the appropriate boundary cellstring and either go to *CMS-FLOW | Assign BC* or right-click on the cellstring and select *Assign BC*.
    4. In the *CMS-FLOW Boundary Conditions* window, select *WSE and Velocity-forcing*.
    5. Click on *Extract* under the *Time Series* section.
    6. In the *Extract Boundary Conditions* window, select the water surface elevation dataset under the section called *Regional scalar model solution:*, and select the current velocity dataset (vector dataset) under the section called *Regional vector model solution:*.
    7. Click on *Extract*. The extraction process may take several minutes. Be patient and DO NOT click or press any keys in SMS as this may cause the interface to crash.

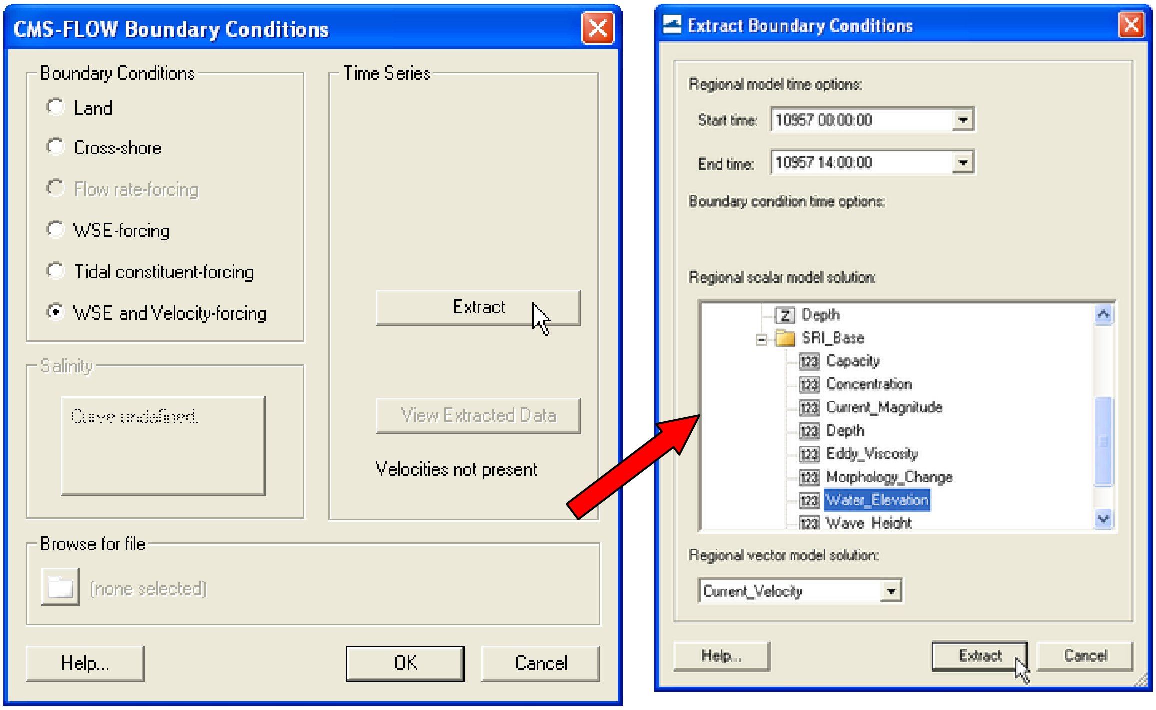

Figure 5.4.6. Assigning water surface elevation and velocity boundary condition.

Cross-shore boundary condition

For a well-defined hydrodynamic problem, the flow discharge or velocity is needed at inflow boundaries, while the water level is usually given at outflow boundaries for a subcritical flow or at inflow boundaries for a supercritical flow. For most coastal applications, the water surface elevation is usually specified at the seaward ocean boundary, which can be the time series of tide levels recorded at the boundary, generated using tidal constituents, or calculated using a larger-scale regional model. Along the cross-shore, constant water surface elevation is not appropriate since winds and waves will produce a setup in the water levels. Also, extracted water levels from a larger grid solution may be too coarse to properly describe the nearshore longshore currents and water levels. A better approach is the use one-dimensional solution to the momentum equations to estimate an inflow water flux and outflow water surface elevation.

- To create a cross-shore boundary condition:

  1. Select the appropriate cellstring and either right-click and select *Assign BC* or go to *CMS-FLOW | Assign BC*
  2. In the *CMS-FLOW Boundary Conditions* window, select *Cross-shore* under *Boundary Conditions* and click *OK*

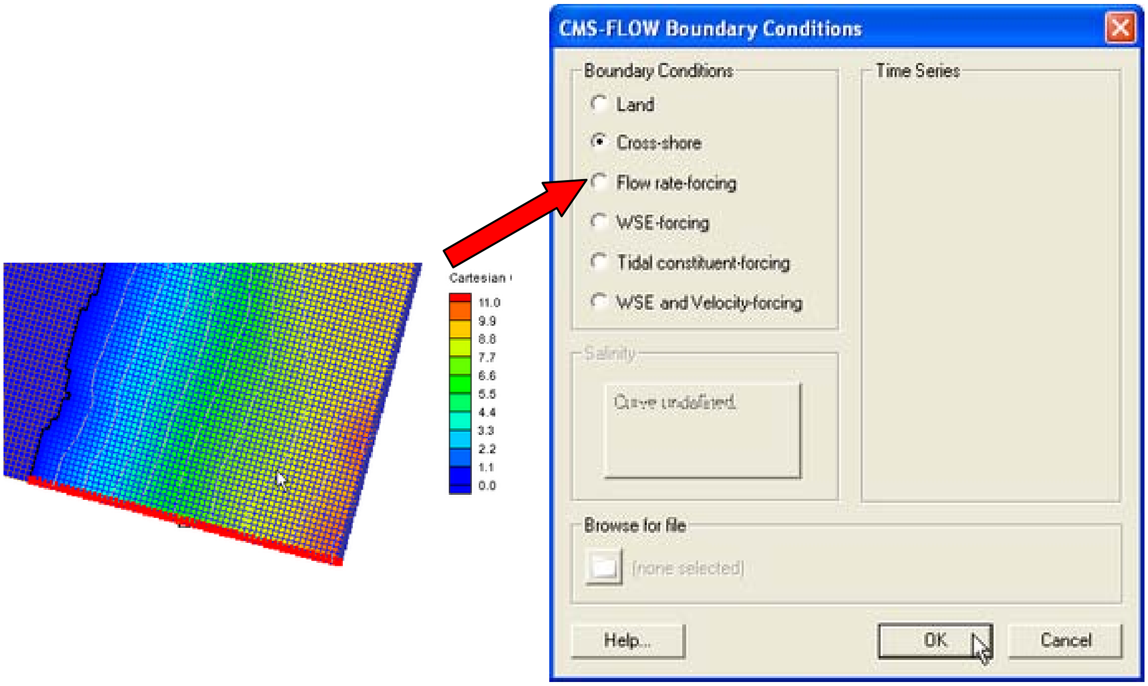

Figure 5.4.7. Assigning a cross-shore boundary condition in SMS 11.0.

5.5 CMS-Flow Sediment Transport

The sediment transport controls are located in the *Transport* section of the *CMS-Flow Model Control* window as shown in the figure below. The sediment transport is activated by going to the *Transport* section of the *CMS-Flow Model Control* and checking the box labeled *Calculate sediment transport*. The CMS card used to turn on or off the sediment transport is described in Table 5.5.1.

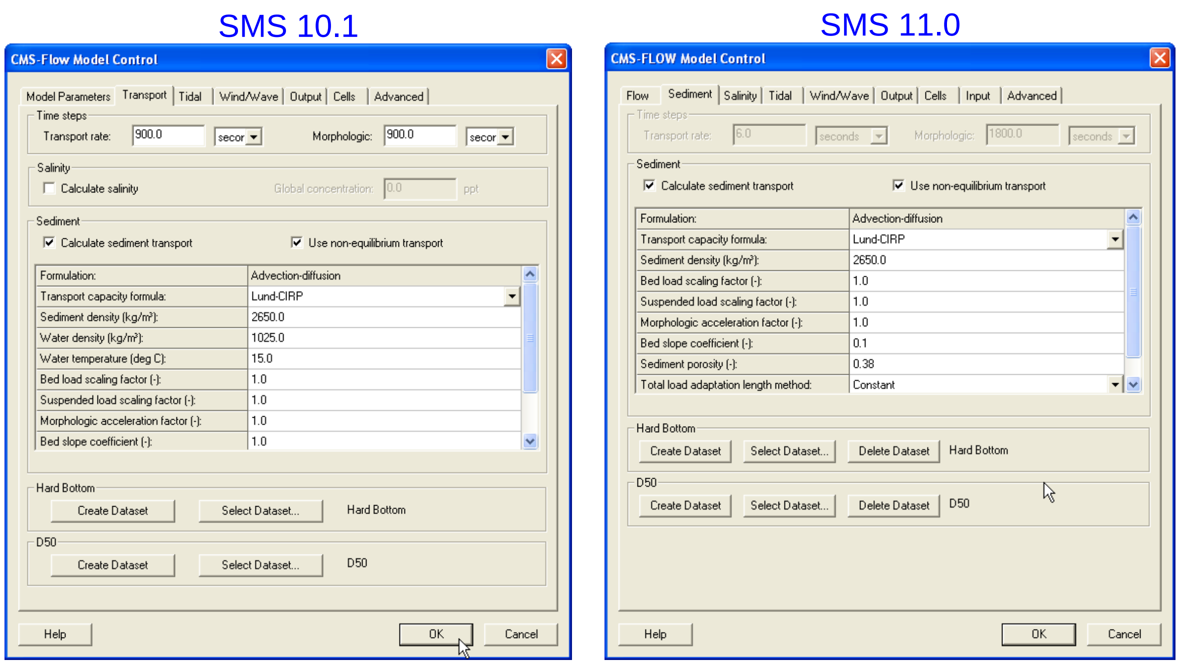

Figure 5.5.1. SMS interface for sediment transport.

Table 5.5.1. CMS card used to turn *On* or *Off* the sediment transport

Card	Arguments	Default	Range	Description
CALC_SEDIMENT_TRANSPORT	CHARACTER	OFF	ON | OFF	Turns on or off the sediment transport calculation.

5.5.1 Transport model

There are presently three sediment transport models available in CMS: (1) Equilibrium total load, (2) Equilibrium bed load plus advection-diffusion for suspended load, and (3) Non-equilibrium total load. The first two models are selected by unchecking the checkbox which says "Use non-equilibrium transport" and selecting either "Total load" for the first model, or "Advection-diffusion" for the second next to input item named "Formulation". The third model is selected by checking the box "Use non-equilibrium transport".

Table 5.5.2. CMS card used to select sediment transport model and time steps for transport equation and updating bed elevation

Card	Arguments	Default	Range	Description	Versions
SED_TRAN_FORMULATION	CHARACTER	NET	WATANABE | LUND_CIRP | A-D | NET	Selects the sediment transport model.	>1.0
SED_TRAN_CALC_INTERVAL	REAL		greater or equal to hydro time step for explicit scheme, or equal hydro time step for implicit scheme	Time step used for transport equation	
MORPH_UPDATE_INTERVAL	REAL		greater or equal to hydro time step for explicit scheme, or equal hydro time step for implicit scheme	Time step used for updating bed elevation	

Important Note: When selecting the equilibrium total load model, the SED_TRAN_FORMULATION card is set to either WATANABE or LUND_CIRP depending on the transport formula chosen. When selecting the equilibrium A-D model, the transport formula is specified through the concentration profile formula (described below).

1. **Equilibrium Total load**

In this model, both the bed load and suspended load are assumed to be in equilibrium. The bed change is solved using a simple mass balance equation known as the Exner equation.

2. **Equilibrium Bed load plus Advection-Diffusion Suspended Load**

Calculations of suspended load and bed load are conducted separately. The bed load is assumed to be in equilibrium and is included in the bed change equation while the suspended load is solved through the solution of an advection-diffusion equation. Actually the advection diffusion equation is a non-equilibrium formulation, but because the bed load is assumed to be in equilibrium, this model is referred to the "Equilibrium A-D" model.

3. **Nonequilibrium Total Load**

In this approach, neither the bed nor suspended loads are assumed to be in equilibrium. The suspended- and bed-load transport equations are combined into a single equation and thus there is one less empirical parameter to estimate (adaptation length).

Important Note: Only the Nonequilibrium Total Load model is available in CMS versions 4.0 and greater at the moment. Plans are under way to include the Equilibrium Bed plus Advection-Diffusion Suspended Load model as an option, but the Equilibrium Total Load model will likely be discontinued.

5.5.2 Transport Formula

The near-bed sediment concentration or concentration capacity are calculated with one of the following transport formula:

1. Lund-CIRP (2006)
2. Van Rijn (1998)
3. Watanabe (1987)
4. Soulsby-van Rijn (1997) ($\geq$V4.0)

Table 5.5.3 illustrates how to set the transport formula.

Table 5.5.3. CMS card used to set the transport formula.

Card	Arguments	Default	Range	Description
NET_TRANSPORT_CAPACITY	CHARACTER	LUND-CIRP	LUND-CIRP | VAN_RIJN | WATANABE | SOULSBY	Selects the transport formula. Note that SOULSBY is only available in v>=4.0
TRANSPORT_FORMULA	CHARACTER	LUND-CIRP	LUND-CIRP | VAN_RIJN | WATANABE | SOULSBY	Selects the transport formula. Note that SOULSBY is only available in v>=4.0.
SED_TRANS_FORMULATION	CHARACTER	LUND-CIRP	LUND-CIRP | A-D | WATANABE | NET	Selects the transport formula for the equilibrium total load model. Does not specify the transport formula for the equilibrium A-D and non-equilibrium total load models.
CONCENTRATION_PROFILE	CHARACTER	LUND-CIRP	LUND-CIRP|EXPONENTIAL| ROUSE| VAN_RIJN	Selects the concentration profile to be used either in the equilibrium A-D or total load nonequilibrium models.
A_COEFFICIENT_WATANABE	REAL	0.1	0.05-0.5	Empirical coefficient which goes into the Watanabe transport formula.

3. **Important Notes:**
   1. Different transport formula may produce very different results in morphology change.
   2. The Lund-CIRP does well in predicting the surf zone sediment transport but tends to overestimate the transport rates near the wetting and drying limit and in deep water (>10 m).
   3. The Soulsby-van Rijn and van Rijn transport formula tends to underestimate the transport for conditions near the critical shear stress of motion. The formula also tends to underestimate the transport close to the shoreline.

5.5.2 Scaling Factors

Transport Scaling Factors

The bed and suspended transport scaling factors multiply the transport capacity or near-bed sediment concentration calculated from the transport formula. These factors should be used to calibrate sediment transport rates and due to the large uncertainty in the transport formula, it is generally acceptable to use scaling factors in the range of 0.5-2.0 (Table 5.5.4) .

Table 5.5.4. CMS card used to define scaling factors

Card	Arguments	Default	Range	Description
BED_LOAD_SCALE_FACTOR	REAL	1.0	0.5-2.0	Calibration factor for bed load transport capacity formula
SUSP_LOAD_SCALE_FACTOR	REAL	1.0	0.5-2.0	Calibration factor for suspended load transport capacity formula

Morphologic Acceleration Factor

The morphologic scaling factor is directly multiplied by the calculated bed change at every time step and is intended as a means of speeding up the computational time. It is only recommended for periodic boundary conditions or conditions that do not change rapidly over time (see Table 5.5.5).

Table 5.5.5. CMS card used to set the morphologic acceleration factor

Card	Arguments	Default	Range	Description
MORPH_ACCEL_FACTOR	REAL	1.0	1-100	Morphologic acceleration factor. Directly multiplies calculated bed change.

- **Note**: The morphologic scaling factor is NOT a calibration parameter. It should only be used in cases with periodic forcing and boundary conditions and even then it should be used with caution. It is NOT recommended to use larger values than 20-30.

Schmidt Number

The sediment mixing coefficient is calculated as the eddy viscosity divided by the Schmidt number. For simplicity the Schmidt number is assumed to be constant and the default value is 1.0 (see Table 5.5.6).

Table 5.5.6. CMS card used to set the Schmidt number

Card	Arguments	Default	Range	Description	Versions
SCHMIDT_NUMBER	REAL	1.0	none	Controls the sediment mixing strength	v>=4.0

- **Note**: The Schmidt number should NOT be used as a calibration number and should only be changed in sensitivity analysis or model testing.

5.5.5. Sediment Characteristics

The sediment characteristics are set in the "Sediment" tab of the "CMS-Flow Model Control" window. The sediment characteristics are the porosity, density, shape, and fall velocity (see Table 5.5.7).

Table 5.5.7. CMS-Flow card for sediment characteristics.

Card	Arguments	Range	Default	Description	Versions
SEDIMENT_POROSITY	REAL	0-1	0.4	Sets the sediment porosity	
SEDIMENT_DENSITY	REAL	none	2650	Sets the sediment density in kg/m^3	
SEDIMENT_FALL_VELOCITY	REAL	4.0e-4 - 0.4	none	Sets the sediment fall velocity to a constant in m/s	v>=3.5
SEDIMENT_FALL_VELOCITIES	INTEGER REAL REAL ...	none	none	Sets the sediment fall velocity to a constant in m/s for multiple grain size classes. The first number is the number of size classes and is followed by the fall velocities for each size class in ascending order.	v>=4.0
SEDIMENT_FALL_VEL_FORM	CHARACTER	SOULSBY-WHITEHOUSE | WU-WANG	SOULSBY-WHITEHOUSE	Sets the sediment fall velocity formula.	v>=4.0
SEDIMENT_COREY_SHAPE_FACTOR	REAL	none	0.7	Sets the Corey shape factor which is used in the Wu-Wang sediment fall velocity formula.	v>=4.0

- **Important Notes:**

  1. It is NOT recommended to use the sediment fall velocity, porosity, density or shape factor as calibration parameters. These parameters should be estimated using field or literature data.
  2. The sediment porosity and density are assumed constant for the whole domain and all grain size classes. For most coastal applications these assumptions are reasonable but need to be taken into consideration.

5.5.6 Bedslope Term

The bedslope term accounts for the effect of gravity on sloped beds. The larger the bed slope coefficient, the more sediment tends to move downslope, thus smoothing the solution. The CMS-Flow used to specify the slope coefficient is described in the table below. The bedslope coefficient is set in the *Sediment* tab of the *CMS-Flow Model Control* window in SMS 11.0.

Table 5.5.8. CMS card used to set the bedslope coefficient.

Card	Arguments	Default	Range	Description	Versions
SLOPE_COEFFICIENT	REAL	1.0	0-5	Bed slope coefficient which controls a diffusion term which moves sediment down slope	

5.5.7 Avalanching

Avalanching is the process of sediment sliding when the critical angle of repose is reached. In CMS, avalanching is simulated using a mass conservative relaxation method which limits the bed slope to the critical angle of repose. For most coastal applications, the critical angle of repose is never reached, so it is not needed. The CMS-Flow cards used for specifying avalanching, and its options, are described in Table 5.5.9.

Table 5.5.9. CMS-Flow cards related to avalanching.

Card	Arguments	Range	Default	Description
USE_AVALANCHING	CHARACTER	ON | OFF	ON	Turns On or Off the avalanching.
RESPOSE_ANGLE	REAL	none	32º	Specifies the angle of repose in degrees.
AVALANCHE_MAX_ITERATIONS	INTEGER	none	200	Specifies the maximum number of iterations used in the implicit solution scheme. For the explicit solution scheme, the avalanching is calculated every transport time step for one iteration.

5.5.8 Ramp Period

The option is available to not calculate the morphology change during the ramp period (Table 5.5.10). The best practice is the start the model simulation so that the time when the ramp period ends corresponds to the time of the measured bathymetry. This avoid the initial bed erosion (although slight) of the bed. This also facilities calculating simulation statistics such as transport rates and residual currents.

Table 5.5.10. CMS-Flow card used to turn *On* or *Off* bed updating during the ramp period.

Card	Arguments	Range	Default	Description	Versions
CALC_MORPH_DURING_RAMP	CHARACTER	ON | OFF	ON	Turns On or Off the morphology change calculation during the ramp period	v>=4.0

5.5.9 Total Load Correction Factor

The total load correction factor accounts for the nonuniform vertical profile of sediment concentration and current velocity and produces temporal lag in between the flow and sediment transport (Table 5.5.11). The factor is used in the nonequilibrium total load sediment transport formula.

Table 5.5.11. CMS-Flow cards related to the total load correction factor.

Card	Arguments	Default	Range	Description
TOTAL_LOAD_CORR_FACTOR_CONSTANT	REAL	0.3-1.0	none	Sets the total load correction factor to a constant.
CONCENTRATION_PROFILE	CHARACTER	LUND-CIRP | VAN_RIJN | EXPONENTIAL | ROUSE	none	Sets the concentration profile to be used either in the pickup and deposition functions or the total load correction factor calculation.

5.5.10 Single-sized Sediment Transport

Hiding and Exposure Correction

This cell-specific parameter allows for the single-sized sediment transport model to make a hiding and exposure correction factor (Table 5.5.12).

Table 5.5.12. CMS card used to set hiding and exposure correction factors

Card	Arguments	Default	Range	Description
TRANSPORT_GRAIN_SIZE	REAL	none	none	Transport grain size in mm. The transport grain size is the sediment size which is eroded, transported, and deposited. When this card is specified the D50 dataset is used to make a hiding and exposure correction to the critical shear stress.
HIDING_EXPOSURE_COEFFICIENT	REAL	0.7	0.6-1.0	Hiding and exposure coefficient.

Important Notes

- If the no transport grain size is specified, then the transport grain size is calculated based on the mean grain size of the whole domain.
- If the card CONSTANT_GRAIN_SIZE is used, then the D50 dataset will be ignored.

Boundary and Initial Conditions

In the case of the Equilibrium Total Load sediment transport model, all boundaries are set to the equilibrium transport rate. For the Equilibrium Bed Load plus Advection Diffusion model, the suspended load is specified as the equilibrium concentration at inflow cells and a zero gradient at outflow cells. For the Total load nonequilibrium sediment transport model, the sediment concentration is set to the equilibrium concentration at inflow cells and a zero gradient boundary condition is applied at outflow cells. In the case an initial conditions file is NOT specified both the hydrodynamics and sediment concentrations are initialized as zero. If an initial conditions file is specified, than the initial sediment concentrations are read in. If an initial conditions file is specified but without the sediment concentration, than the initial sediment concentration is set to the equilibrium concentration.

Table 5.5.13. CMS card used to set the sediment boundary and initial conditions

Card	Arguments	Default	Range	Description	Versions
NET_LOADING_FACTOR	REAL	1.0	0.5-2.0	Used to specify under- or overloading at sediment inflow boundaries. Only for NET.	3.5=>v<=4.0
SEDIMENT_INFLOW_LOADING_FACTOR	REAL	1.0	0.5-2.0	Used to specify under- or overloading at sediment inflow boundaries.	>=4.0
CALC_MORPH_DURING_RAMP	CHARACTER	ON	ON | OFF	Determines whether to calculate the morphology change during the ramp period	v>=3.5

5.5.11 Multiple-sized Sediment Transport

The CMS multiple-sized sediment transport model is calculated using the multifraction approach, in which the total sediment transport is equal to the sum of the transports of discrete sediment sizes classes.

The input information for multiple-sized sediment transport can be divided into the following groups:

1. Grain size classes
2. Fractional bed composition
3. Bed layer thickness
4. Mixing layer
5. Hiding and Exposure

Further details on each type of information is provided in the sections below.

Grain Size Specification

In the multiple-sized sediment transport model, the continuous grain sizes are divided into discrete bins or size classes. Each size class has a character diameter, and lower and upper limits. Because sediment sizes classes are transport from one cell to another the size classes must be constant over the whole domain. In CMS there are two options for specifying the grain size information:

1. Characteristic diameters
2. Size class limits

The characteristic diameters d_k are calculated using the geometric mean of the lower $d_{l,k}$ and upper $d_{u,k}$ bin limits

$$d_k = \sqrt{d_{l,k} d_{u,k}} \tag{5.5.1}$$

The bin width is given by

$$\Delta d_k = d_{u,k} - d_{l,k} \tag{5.5.2}$$

Table 5.5.14 describes the CMS-Flow cards related to the grain size class information.

Table 5.5.14. CMS-Flow cards related to the grain size classes

Card	Arguments	Example	Description
SEDIMENT_SIZE_CLASS_NUMBER	INTEGER	3	Specifies the number of grain size classes.
MULTIPLE_GRAIN_SIZES	INTEGER [REAL, REAL,..., REAL]	3 0.1 0.2 0.3	Specifies the number of grain sizes followed by the grain sizes for each size class in mm.
SEDIMENT_SIZE_CLASS_DIAMETERS	INTEGER [REAL, REAL,..., REAL]	3 0.1 0.2 0.3	Specifies the number of grain sizes followed by the grain sizes for each size class in mm. Same as MULTIPLE_GRAIN_SIZES.
SEDIMENT_SIZE_CLASS_LIMITS	INTEGER [REAL, REAL,..., REAL]	4 0.07 0.15 0.26 0.34	Specifies the size classes limits. The first entry is the number of size class limits which is one greater than the number of size classes.

Notes:

- It is only necessary to specify either the size class characteristic diameters or limits. The other will be calculated internally by the model.

Recommendations:

- Because each grain size requires the iterative solution of a separate transport formula, it is recommended to not use more 8-9 sediment size classes. For most cases, 3-5 sediment size classes are sufficient.
- A good and simple way of estimating the grain size classed based only on the size limits of the distribution is by distributing the diameters logarithmically so that

$$d_k = \exp\left[\ln d_1 + \ln\left(d_N/d_1\right)\frac{k-1}{N-1}\right] \tag{5.5.3}$$

Matlab Example

In order to determine the appropriate grain sizes for a simulation it is useful to be able to determine ahead of time the size class fractions using a log-normal distribution for different median grain sizes and sorting. The Matlab example below determines the grain size distribution given the smallest and largest grain sizes, number of sediment sizes, median grain size and geometric standard deviation. The figure shows the computed grain size distribution.

```
clear all; close all
%--- Start Input ---
d1 = 0.234;  %mm, smallest grain size
dn = 2;      %mm, largest grain size
nsed = 5;    %number of grain sizes
d50 = 0.4;   %mm, median grain size
sg = 1.5;   %mm, geometric standard deviation
%--- End Input ---
%Characteristic diameters
d = exp(log(d1) + log(dn/d1)*((1:nsed)-1.0)/(nsed-1));
%Limits or bounds
dlim = zeros(1,nsed+1);
dlim(2:nsed)=sqrt(d(2:nsed).*d(1:nsed-1));
dlim(1)=d(1)*d(1)/dlim(2);
dlim(nsed+1)=d(nsed)*d(nsed)/dlim(nsed);
%Fractions
p = diff(dlim).*lognpdf(d,log(d50),log(sg));
p = p/sum(p);
%Plotting
figure
hold on
for k=1:nsed
  fill([dlim(k) dlim(k+1) dlim(k+1) dlim(k)],...
    [0 0 p(k) p(k)]*100,0.5*[1,1,1])
end
ylabel('Fraction, %')
xlabel('Grain size, mm')
set(gca,'box','On','TickDir','out',...
  'XMinorTick','OFF','YMinorTick','OFF')
[d',p']
return
```

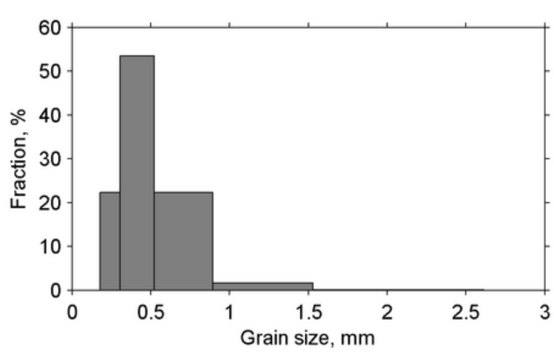

Figure 5.5.2. Computed grain size distribution from the Matlab script above.

Fractional Bed Composition

Since the sediment size classes are constant for the whole domain, the bed composition is specified as the fractional amount of each size class per cell and per bed layer. In CMS V4.0 there are 5 options for specifying the fractional bed composition:

1. Two-dimensional horizontal (2DH) D50 dataset plus spatially constant geometric standard deviation (option D50_SIGMA)
2. 2DH D16, D50, and D84 datasets (option D16_D50_D84)
3. 2DH D35, D50, and D90 datasets (option D35_D50_D90)
4. Spatially (horizontally and vertically) constant size class fractions (option SIZE_CLASS_FRACTIONS)
5. Fraction of each sediment size class for each cell and bed layer (option FRACTIONS_DATASET)

A description of the CMS-flow card used to select each bed material composition option is described in Table 5.5.15.

Table 5.5.15. CMS-Flow card used for specifying the bed material composition

Card	Arguments	Default	Range	Description
BED_COMPOSITION_INPUT	CHARACTER	none	D50_SIGMA | D16_D50_D84 | D35_D50_D90 | SIZE_CLASS_FRACTIONS | FRACTIONS_DATASET	Selects the method for specifying the bed material composition.

Further details on each option is provided in the sections below.

D50 Dataset and Standard Deviation

This is the simplest and easiest way to specify the bed composition for multiple-sized sediment transport in CMS because it only requires the D50 dataset which is already in the SMS interface and one extra parameter, the geometric standard deviation.

- **Assumptions:**
  1. The initial sediment sorting is constant for the whole domain
  2. The initial bed composition may vary spatially according to the D50 dataset but is constant with depth
  3. The initial grain size distribution can be approximated by a log-normal distribution

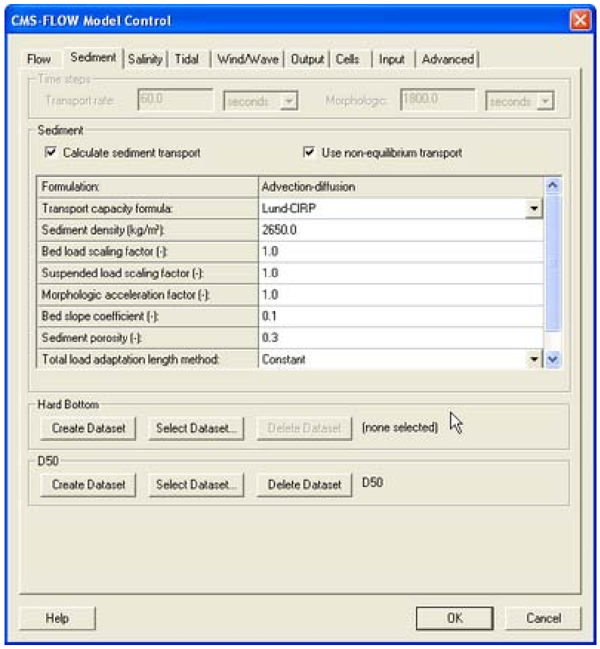

Figure 5.5.3. ***Sediment*** **tab of the** ***CMS-Flow Model Control*** **window (SMS 11.0)**

The geometric standard deviation σ_g can be defined using the method of moments

$$\sigma_g = \exp\sqrt{\sum p_k(\ln d_k - \ln d_g)^2} \tag{5.5.4}$$

where p_k is the fraction, d_k is the size class diameters in mm, d_g is the geometric mean in mm given by

$$d_g = \exp\sum p_k \ln d_k \tag{5.5.5}$$

An alternate approach to Equation 5.5.4 for obtaining geometric standard deviation is by using the graphical measures (cumulative percentile values) (Folk and Ward, 1957)

$$\sigma_g \approx \sigma_G = \exp\left(\frac{\ln d_{16} - \ln d_{84}}{4} + \frac{\ln d_5 - \ln d_{95}}{6.6}\right) \approx \left(\frac{d_{16}}{d_{84}}\right)^{1/2} \tag{5.5.6}$$

where diameter subscript indicates the percentile value. The size class fractions p_k are calculated using log-normal sediment size distribution

$$p_k = \frac{\Delta d_k}{d_k \ln\sigma_g\sqrt{2\pi}}\exp\left[-\frac{1}{2}\left(\frac{\ln d_k - \ln d_{50}}{\ln\sigma_g}\right)^2\right] \tag{5.5.7}$$

Table 5.5.16. Sorting classification based on the geometric standard deviation (Folk and Ward, 1957)

Geometric Standard Deviation	Sorting Classification
<1.27	Very well sorted
1.27-1.41	Well sorted
1.41-1.62	Moderately well sorted
1.62-2.00	Moderately sorted
2.00-4.00	Poorly sorted
4.00-16.00	Very poorly sorted
>16.00	Extremely poorly sorted

A description of the CMS-Flow cards related to the D50_SIGMA option for specifying the bed material composition is provided in table 5.5.17.

Table 5.5.17. CMS-Flow cards related to the D50 dataset and standard deviation

Card	Arguments	Default/Format	Description
D50_DATASET	CHARACTER CHARACTER | CHARACTER	[file name] [dataset path and name]	Specifies the D50 dataset file followed by the path and name. If the file name is not specified than it is assumed to be the grid file (*_grid.h5).
SEDIMENT_STANDARD_DEVIATION	REAL	1.5	Specifies the geometric sediment standard deviation for all cells and layers in mm.

D16, D50, and D84 Datasets

The second approach for specifying the initial bed composition is by using D16, D50, and D84 datasets. Because only the D50 dataset can be specified through the SMS interface, it is necessary to specify the additional datasets either through the *Custom Input Datasets* are by manually creating the datasets using the *Data Calculator* and exporting them to XMDF file(s).

- **Assumptions:**
  1. The initial bed composition may vary spatially according to the D16, D50, and D84 datasets but is constant with depth
  2. The initial grain size distribution can be approximated by a log-normal distribution

The bed sorting is estimated as

$$\sigma_g \approx \left(\frac{d_{16}}{d_{84}}\right)^{1/2} \tag{5.5.8}$$

The fractional bed composition is calculated using a log-normal distribution according to Equation 5.5.7. Table 5.5.18describes the CMS-Flow cards used for specifying the bed material composition using the D16, D50, and D84 datasets.

Table 5.5.18. CMS-Flow cards related to the D16_D50_D84 datasets

Card	Arguments	Default/Format	Description
D16_DATASET	CHARACTER CHARACTER	[file name] [dataset path and name]	Specifies the D16 dataset path and file name.
D50_DATASET	CHARACTER CHARACTER | CHARACTER	[dataset path and name] | [file name] [dataset path and name]	Specifies the D50 dataset path and file name. If the file is not specified than it is assumed to be the model grid file (*_grid.h5).
D84_DATASET	CHARACTER CHARACTER	[file name] [dataset path and name]	Specifies the D84dataset path and file name.
CUSTOM_DATASET	CHARACTER CHARACTER	[file name] [dataset path and name]	Specifies a custom dataset path and file name. The custom dataset can be used for any additional user defined dataset. The name of the dataset must be equal to the variable name. For example the D16 dataset must be named D16 and cannot be myD16.

The first approach of specifying the D16 and D84 datasets is by using the *Custom Input Datasets* section under the *Input* tab of the *CMS-Flow Model Control* window. The advantage of using this approach is that the user can edit the datasets once they are created without having to map each dataset as the *Elevation* dataset. It also avoids having to specify the dataset files and names manually in the *.cmcards file. The steps for creating and specifying the D16 and D84 datasets are outlined below:

- **Custom Datasets**

1. Open *CMS-Flow Model Control* window and go to the *Input* tab.

2. To create the D16 dataset, click on the *New* buttom and a window will appear titled *New Editable Dataset* (Figure 5.5.4).

3. Name the dataset D16 for the D16 dataset or D84 for the D84 dataset. The names must be correct for CMS to be able to interpret the dataset.

4. Set the initial values by either specifying a spatially constant value or selecting a current dataset. Click *OK* in the *New Editable Dataset* window

5. Repeat steps 2 through 5 for the D84 dataset. Click *OK* in the *CMS-Flow Model Control* window. Save the CMS-Flow project

6. Once the datasets are created the SMS project explorer will show a folder with the D16 and D84 datasets (Figure 5.5.5).

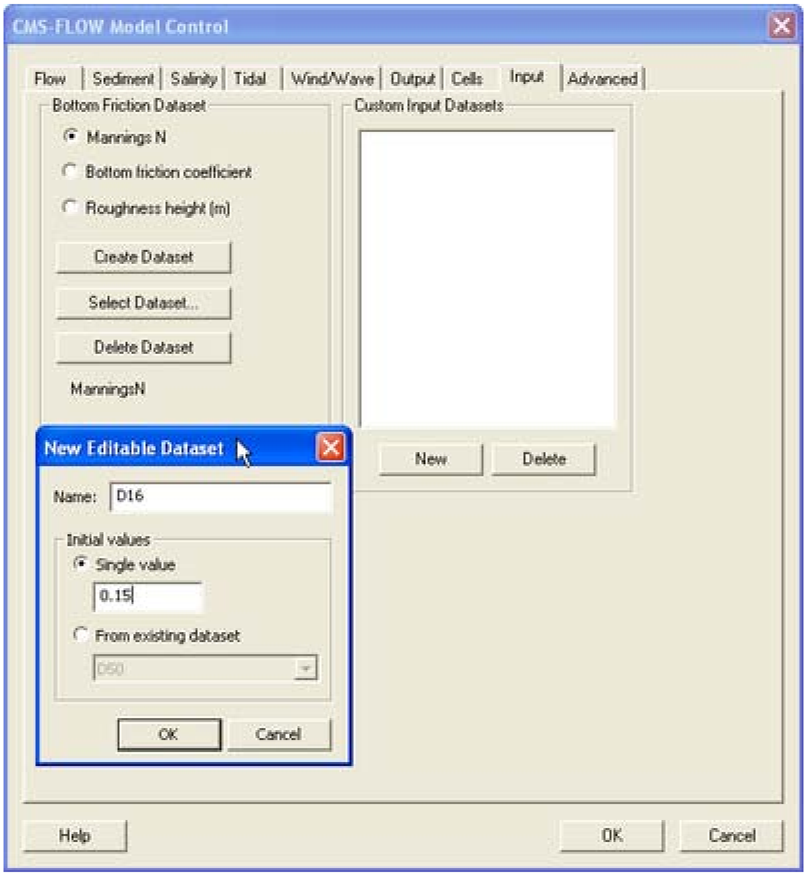

Figure 5.5.4. SMS project explorer showing Custom Datasets for D16 and D84.

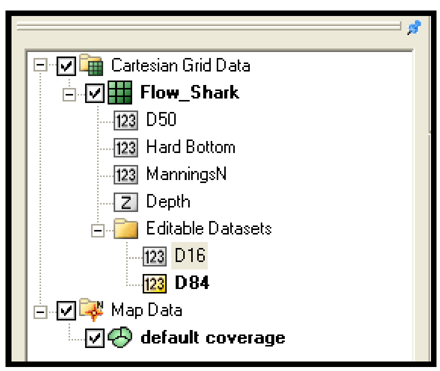

Figure 5.5.5. SMS project explorer showing Custom Datasets for D16 and D84.

7. After saving the CMS-Flow project the project folder will show a new dataset called *Flow_Shark_input_datasets.h5* (Figure 5.5.6).
8. The data structure of the *Flow_Shark_input_datasets.h5* file is shown in Figure5.5.7.

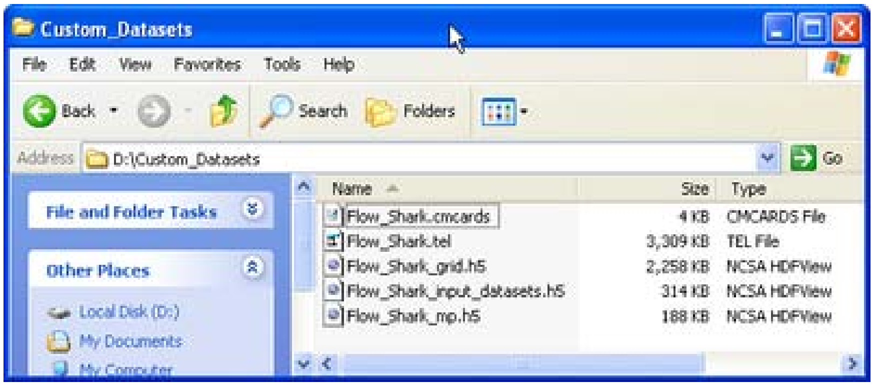

Figure 5.5.6. SMS project explorer showing Custom Datasets for D16 and D84.

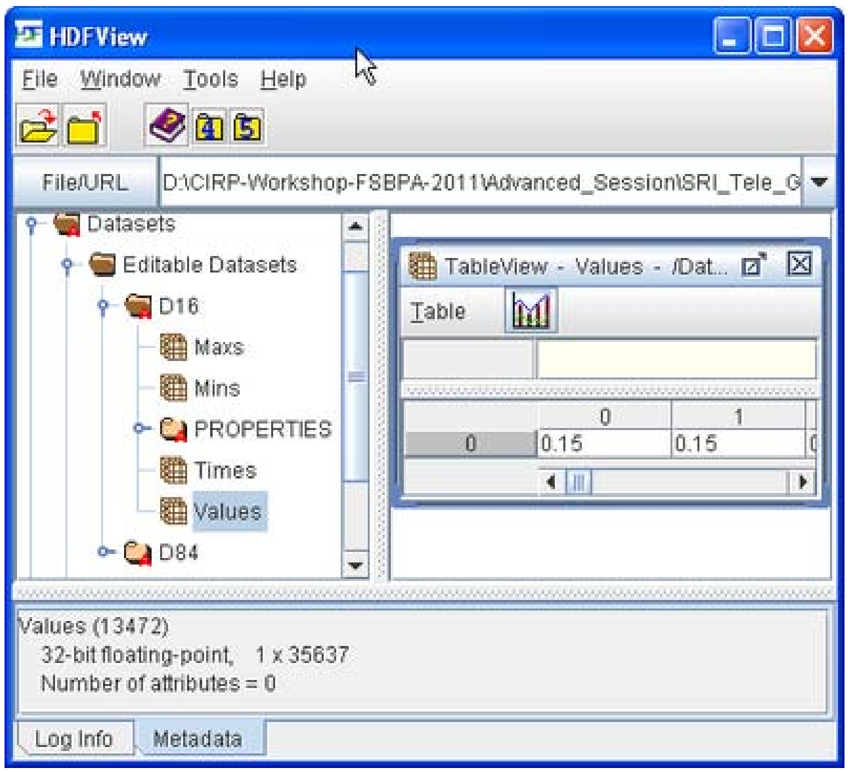

Figure 5.5.7. Structure of the *Custom Input Datasets* file shown in XMDFView.

An alternate approach for specifying datasets is by exporting them into XMDF files and then specifying the file names and paths in the *Advanced* cards section of the *.cmcards file. The advantage of this approach is that is allows more flexibility in specifying project alternatives and also works for SMS versions 10.1 and earlier. To export the D16 and D84 datasets

- **Exported Datasets**

1. Create and initialize the D16 and D84 datasets by clicking on the menu *Data* | *Data Calculator* (Figure 5.5.8). Enter a constant value under the *Calculator* section and make sure the *Output dataset name* is either D16 or D84.

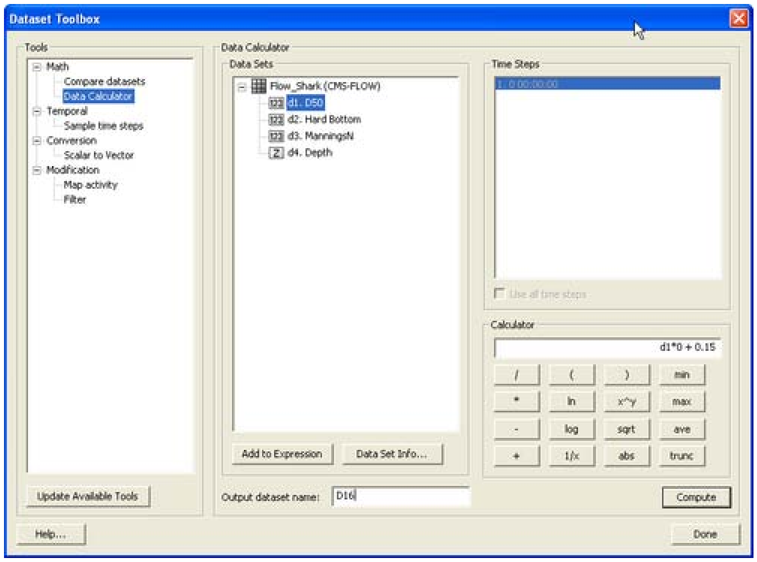

Figure 5.5.8. SMS project explorer showing Custom Datasets for D16 and D84.

If necessary, edit the D16 and D84 datasets. As shown in Figure 5.5.9, it is not possible to edit the D16 and D84 datasets once they are created. A workaround for this is to set either dataset as the *Elevation* dataset, which then allows you to edit the dataset.

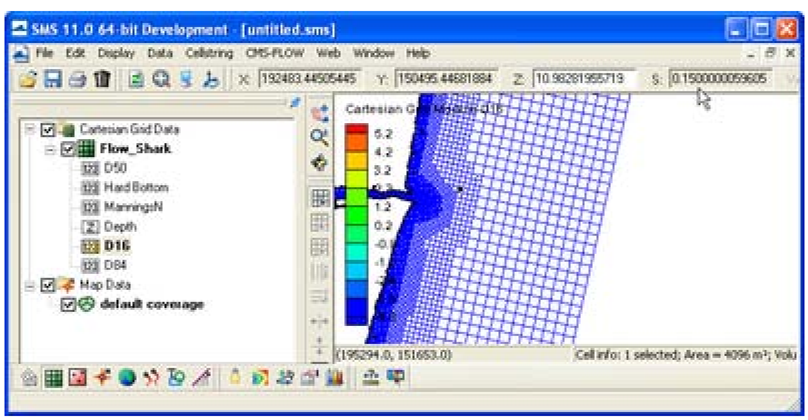

Figure 5.5.9. It is not possible to edit user defined datasets unless they are the Depth, D50, ManningN, Hardbottom, or a Custom Dataset.

2. Set the D16 dataset as the *Elevation* dataset: Click on *Data* | *Map elevation* (Figure 5.5.10). A window titled Select Dataset will appear. Select the D16 dataset and click Select.

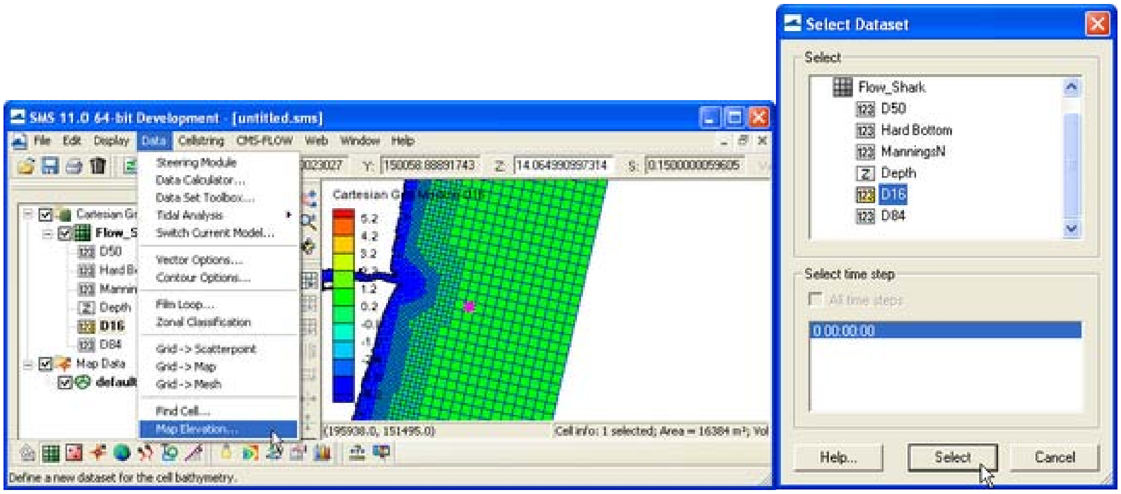

Figure 5.5.10. Mapping of D16 dataset as the *Elevation* dataset (left). *Elevation* dataset selection (right).

3. The D16 dataset will show with a large *Z* on the left indicating that it is mapped as the *Elevation* dataset (Figure 5.5.11).

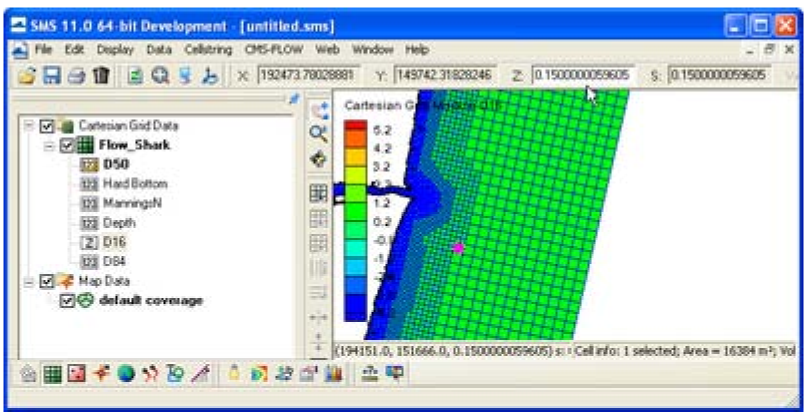

Figure 5.5.11. SMS window showing D16 dataset mapped as the *Elevation* dataset.

4. Once you are finished editing the dataset, make sure to remap the appropriate dataset as the *Elevation* dataset (usually called *Depth* dataset).

5. Export the D16 and D84 datasets to XMDF file(s). The datasets can be exported to the same or separate file (Figure 5.5.12). The important thing is that the correct file name is specified in the Advanced Cards.

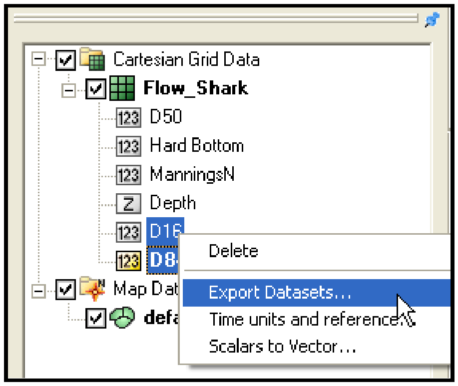

Figure 5.5.12. Exporting the D16 and D84 datasets.

D35, D50, and D90 Datasets

This method is very similar to the previous method. The only difference is that the D35 and D90 datasets are used to estimate the bed sorting instead of the more commonly used D16 and D84 datasets. The reason for including this method is that the D35 and D90 datasets are also required for running the Particle Tracking Model so it reduces the number of datasets that need to be prepared by the user in cases where both models are being run.

- **Assumptions:**
  1. The initial bed composition may vary spatially according to the D35, D50, and D90 datasets but is constant with depth
  2. The initial grain size distribution can be approximated by a log-normal distribution

The geometric standard deviation is estimated as

$$\sigma_g \approx \left(\frac{d_{90}}{d_{35}} \right)^{0.61} \tag{3.5.9}$$

The fractional bed composition is then calculated with log-normal distribution according to Equation 5.5.7.

Table 5.5.19 describes the CMS-Flow cards used for specifying the bed material composition using D35, D50, and D90 datasets.

Table 5.5.19. CMS-Flow cards related to the D35_D50_D90 datasets

Card	Arguments	Default/Format	Description
D35_DATASET	CHARACTER CHARACTER	[file name] [dataset path and name]	Specifies the D35 dataset path and file name.
D50_DATASET	CHARACTER CHARACTER | CHARACTER	[dataset path and name] | [file name] [dataset path and name]	Specifies the D50 dataset path and file name. If the file is not specified than it is assumed to be the model grid file (*_grid.h5).
D90_DATASET	CHARACTER CHARACTER	[file name] [dataset path and name]	Specifies the D84dataset path and file name.
CUSTOM_DATASET	CHARACTER CHARACTER	[file name] [dataset path and name]	Specifies a custom dataset path and file name. The custom dataset can be used for any additional user defined dataset. The name of the dataset must be equal to the variable name. For example the D35 dataset must be named D35 and cannot be myD35.

Size Class Fractions

For this case, the fractional bed compositions are specified for each size class for all cells and all bed layers. The fractional bed compositions must sum to 1.0. This option is useful for simulating cases where the bed composition is well known and is constant both in the horizontal and vertical directions.

- **Assumption:**
  1. The initial bed composition is constant at every cell and layer for whole domain

Table 5.5.20. CMS-Flow cards related to the SIZE_CLASS_FRACTIONS option for specifying the bed material composition

Card	Arguments	Example	Description
SEDIMENT_SIZE_CLASS_FRACTIONS	INTEGER [REAL, REAL,..., REAL]	3 0.2 0.5 0.3	Specifies the fractional composition for each size class and for all cells and bed layers. The first entry is the number of grain size classes.

Bed Layer Thickness

The initial bed layer thickness needs to be specified by the user. There are three methods for specifying the initial bed layer thickness:

1. CONSTANT – User specified constant for all cells and bed layers.
2. LAYER – User specified bed layer thickness for each bed layer and is applied to all cells on the grid.
3. LAYER_DATASET – User specified dataset for each cell and bed layer (3D dataset).

The CMS-Flow cards related to the bed layer specification are described in the table below.

Table 5.5.21. CMS-Flow cards related to the bed thickness general parameters

Card	Arguments	Default/Format	Range	Description
BED_LAYER_THICKNESS_INPUT	CHARACTER	none	CONSTANT | LAYER_THICKNESS | LAYER_DATASET	Sets the method for specifying the sediment bed layer thickness.
BED_LAYER_THICKNESS_DATASET	CHARACTER CHARACTER	none	[file] [path]	Specifies the bed layer thickness file and dataset path.
BED_LAYERS_MAX_NUMBER	INTEGER	8	>= Input bed layers	Sets the maximum number of bed layers. M
BED_LAYERS_CONSTANT_THICKNESS	REAL	none	> minimum layer thickness.	Sets the initial bed layer thickness to a constant in meters for all cells and bed layers.
BED_LAYERS_THICKNESS	INTEGER [REAL, REAL,..., REAL]	[N_lay] [thickness 1] ... [thickness N_lay]	none	Sets the thickness for each layer in meters for all cells.
BED_LAYER_MIN_THICKNESS	REAL	0.01	none	Sets the minimum layer thickness. Applicable to all bed layers including the mixing layer (layer 1).
BED_LAYER_MAX_THICKNESS	REAL	0.5	none	Sets the MAXIMUMlayer thickness. Applicable to all bed layers including the mixing layer (layer 1).

- **Important Note:** If the initial bed layer thickness is not specified than it will be set to zero.

Mixing Layer

The mixing layer is the first layer from the surface which is allowed to exchange (or mix) sediments with the sediment transport. The mixing layer thickness is calculated based on the median grain size and bed form size. The option is also available to set the mixing layer to a constant. The CMS-Flow cards used for setting the mixing layer thickness to a constant is described in Table 5.5.22.

Table 5.5.22. CMS-Flow used for setting a constant mixing layer thickness.

Card	Arguments	Default	Range	Description
MIXING_LAYER_CONSTANT_THICKNESS	REAL	none	greater than minimum thickness	Sets the mixing layer thickness to a constant in meters.

- **Notes:**
    1. Using a larger mixing layer will slow down the temporal change in bed material composition and enhance model stability.
    2. Erosional cases tend to have smaller mixing layer thicknesses and depositional cases tend to have larger mixing layer thicknesses.
    3. The mixing layer thickness should NOT be used as a calibration parameter.

Hiding and Exposure

Hiding and exposure is an important parameter for multiple-sized sediment transport. Presently, there are no well established coastal sediment transport formulas for nonuniform sediments. The most common approach is to adapt single-size transport formulas by applying correction factors. The most common of these factors is the hiding and exposure correction. The implementation of this factor varies depending on the transport formula and usually involves a coefficient (exponent) which needs to be calibrated. Because all of the transport formulas used in CMS were originally developed for uniform sediments, there is still a large uncertainty in the value of the hiding and exposure coefficient. It is expected that the hiding and exposure coefficient will be different for different transport formula especially since the implementation of the hiding and exposure correction will vary by the formula that is applied (Table 5.5.23).

Table 5.5.23. CMS-Flow card used for setting the hiding and exposure coefficient.

Card	Arguments	Default/Format	Range	Description
HIDING_EXPOSURE_COEFFICIENT	REAL	1.0	0.4-1.0	Sets the hiding and exposure coefficient.

Examples

- Example 1: The first example used 6 sediment size classes. The size classes are specified with MULTIPLE_GRAIN_SIZES card. The bed composition is specified using the D50_SIGMA method, which requires a geometric standard which is set to 1.3 using the card SEDIMENT_STANDARD_DEVIATION. The initial bed layer thickness is set to a constant 0.5 m.

```
SEDIMENT_MAX_ITERATIONS              30
SEDIMENT_STATISTICS                 120.0  720.0
BED_COMPOSITION_INPUT               D50_SIGMA
MULTIPLE_GRAIN_SIZES                6 0.1 0.126 0.16 0.2 0.25 0.31
SEDIMENT_STANDARD_DEVIATION         1.3
BED_LAYER_CONSTANT_THICKNESS        0.5
MIXING_LAYER_CONSTANT_THICKNESS     0.1
MIXING_LAYER_MIN_THICKNESS          0.05
SEDMIX_OUT_TIMES_LIST               2
BED_OUT_TIMES_LIST                   2
CALC_MORPH_DURING_RAMP              OFF
```

- Example 2: The second example consists of 5 sediment size classes. Note that an alternate card for specifying the sediment size class diameters is used compared to Example 1. The bed material composition is specified with D16, D50, and D84 datasets in separate files.

```
SEDIMENT_MAX_ITER                   30
BED_COMPOSITION_INPUT               D16_D50_D84
D16_DATASET                         "Alt1_D16.h5" "Datasets/D16"
D84_DATASET                         "Alt1_D84.h5" "Datasets/D84"
SEDIMENT_SIZE_CLASS_DIAMETERS       5 0.08 0.12 0.17 0.28 0.45
BED_LAYER_CONSTANT_THICKNESS        0.5
MIXING_LAYER_CONSTANT_THICKNESS     0.02
```

5.6 CMS Surface Roller

The surface roller model is run after each CMS-Wave and before each CMS-Flow run. It runs on the wave grid but is a separate model and module within the CMS. The surface roller model parameters are set within the CMS-Flow cmcards file for convenience.

Table 5.6.1. CMS-Flow cards related to the surface roller

Card	Arguments	Range	Default	Description
CALC_ROLLER	CHARACTER	ON | OFF	OFF	Turns on or off the surface roller model in CMS.
ROLLER_DISSIPATION_COEFFICIENT	REA	0.1	0.05-0.15	Roller dissipation coefficient.
ROLLER_EFFICIENCY_COEFFICIENT	REAL	1.0	0.5-1.0	Roller efficiency coefficient.
ROLLER_SCHEME	CHARACTER	UPWIND1 |UPWIND2 | LAX	UPWIND1	Determines the numerical scheme used for the surface roller calculation. UPWIND1 is a first order upwind scheme, UPWIND2 is a second order upwind scheme, and LAX is the second order Lax scheme.

5.7 Global Output

Global output refers to the variables that are output on every active cell on the grid. The global output options are specified in *Output* tab of the *CMS-Flow Model Control* window. More information on the global output variables, groups and CMS-Flow cards is provided in the sections below. Figure 5.7.1 illustrates the different output datasets for the CMS.

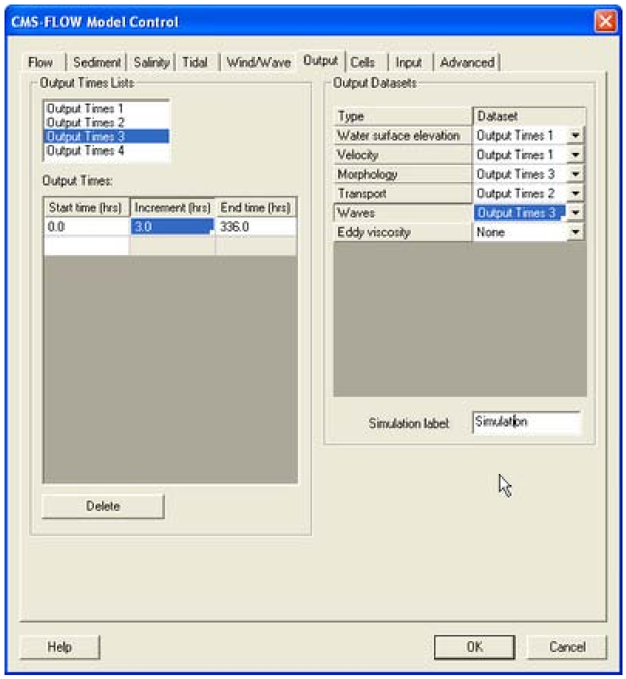

Figure 5.7.1. *Output* tab in SMS 11.0

5.7.1 Output Datasets

A description of the CMS-Flow cards used to specify the global output variable datasets is provided below in Table 5.7.1.

Table 5.7.1. CMS-Flow cards related to the output datasets.

Output Dataset	Group	Description	Units
Current_Velocity	Velocity	Depth-averaged and cell-centered current velocity vector dataset and with respect to local grid coordinates	m/s
Current_Magnitude	Velocity	Depth-averaged and cell-centered current velocity magnitude dataset	m/s
Water_Elevation	Water surface elevaation	Cell-centered water surface elevation	m
Eddy_Viscosity	Eddy viscosity	Cell-centered horizontal eddy viscosity	m^2/s
Concentration	Transport	epth-averaged and cell-centered sediment concentration	kg/m^3

Output Dataset	Group	Description	Units
Capacity	Transport	Depth-averaged and cell-centered sediment concentration capacity	kg/m^3
Total_Sediment_Transport	Transport	Depth-averaged and cell-centered total-load sediment transport	m^2/s
Salinity	Transport	Depth-averaged and cell-centered sediment concentration capacity	kg/m^3
Depth	Morphology	Cell-centered still water depth	m
Morphology_Change	Morphology	Cell-centered morphology (bed) change. Positive is accretion and negative is erosion	m
Wave_Height	Waves	Cell-centered significant wave height	m
Wave_Height_Vec	Waves	Cell-centered significant wave height vector	m
Wave_Period	Waves	Cell-centered peak wave period	s

5.7.2 Output Time Series and Lists

The times at which each group is output is determined by the selecting one of four user defined output time series or lists. In SMS versions 10.1 and earlier, the output time series were used. However, because the output time series can become very large for long-term simulations, the time series have been replaced by lists in which the output times are specifying a list of starting, ending and increments. This option is more compact and also makes it easier to manually change the output options in the cmcards file.

Table 5.7.2. CMS-Flow cards used for specifying the time series and lists.

Card	Aguments/Format	Default value	Description
TIME_SERIES_1	[length of list 1] [output times for list 1]	0	Output time series for list 1 in hours.
TIME_SERIES_2	[length of list 2] [output times for list 2]	0	Output time series for list 2 in hours.
TIME_SERIES_3	[length of list 3] [output times for list 3]	0	Output time series for list 3 in hours.
TIME_SERIES_4	[length of list 4] [output times for list 4]	0	Output time series for list 4 in hours.
TIME_LIST_1	[number of sublists] [sublist 1: start, end, increment] [sublist 2: start, end, increment]...	0	Sublist(s) for output time series 1. For each sublist, the arguments are starting time, end time and increment in hours.
TIME_LIST_2	[number of sublist] [sublist 1: start, end, increment] [sublist 2: start, end, increment]...	0	Sublist(s) for output time series 2. For each sublist, the arguments are starting time, end time and increment in hours.
TIME_LIST_3	[number of sublist] [sublist 1: start, end, increment] [sublist 2: start, end, increment]...	0	Sublist(s) for output time series 3. For each sublist, the arguments are starting time, end time and increment in hours.
TIME_LIST_4	[number of sublist] [sublist 1: start, end, increment] [sublist 2: start, end, increment]...	0	Sublist(s) for output time series 4. For each sublist, the arguments are starting time, end time and increment in hours..

5.7.3 Advanced Output

In addition to the variables specified in the SMS interface, CMS has the option to output advanced mode output including the bed shear stress, bed composition, wind speed, etc.

Table 5.7.3. Advanced output datasets.

Card	Arguments	Default value	Description
WIND_OUT_TIMES_LIST	INTEGER	0	Output time series id for wind velocity and magnitude in m/s.
STRESS_OUT_TIMES_LIST	INTEGER	0	Output time series id for mean bed shear stress in Pa.
WAVE_OUTPUT_DETAILS	ON | OFF	OFF	Outputs additional wave variables including wave direction, radiation stresses, breaking dissipation and roller energy.

5.7.4 XMDF File Compression

The standard CMS-Flow output is written to an XMDF file with the name <Case Name>_sol.h5. The binary file may be written in compressed format using the card described in the table below.

Table 5.7.4. CMS-Flow card for compressing the XMDF output file

Card	Arguments	Default value	Description
XMDF_COMPRESSION	ON | OFF	OFF	Compresses the h5 file by a factor of about 7

5.7.5 Output Cells

Time series at selected *Observation* cells is set in the *Cells* tab of the *CMS-Flow Model Control* window (Figure 5.7.2). A description of CMS-Flow cards used for specifying *Observational* cells are described in Table 5.7.5.

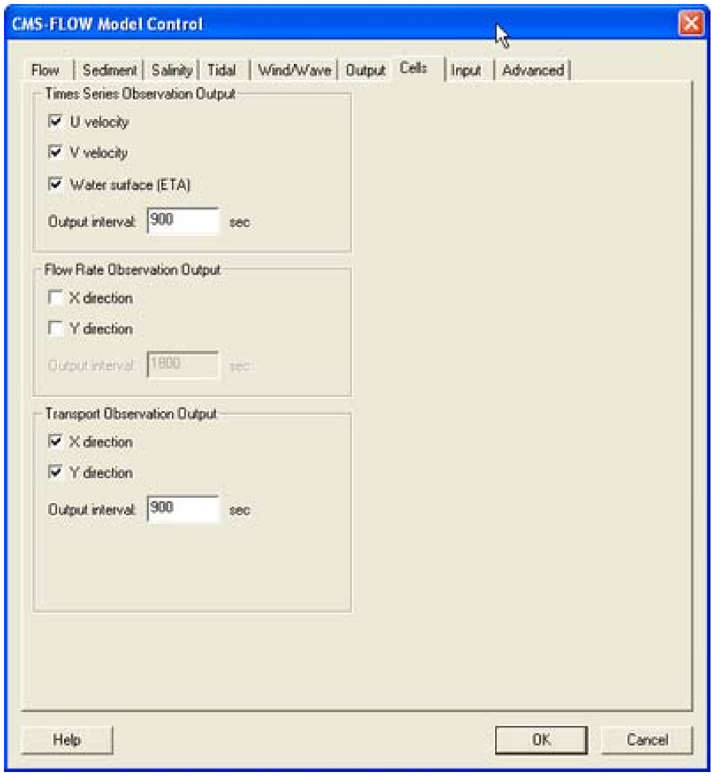

Figure 5.7.2. *Cells* tab in SMS 11.0

Table 5.7.5. CMS-Flow card for compressing the XMDF output file

Card	Arguments	Description
TIME_SERIES_INCREMENT	REAL	Sets the output time increment for the *Time series Observation* points.
ELEV_OBS_CELLS_BEGIN	none	Marks the begging of an *Times series Observation* point list.
ELEV_OBS_CELLS_END	none	Marks the end of an *Times series Observation* point list.
FLOW_RATE_INCREMENT	REAL	Sets the output time increment for the *Flow rate Observation* points.
FLOW_OBS_CELLS_BEGIN	none	Marks the end of an *Flow rate Observation* point list.
FLOW_OBS_CELLS_END	none	Marks the end of an *Flow rate Observation* point list.
Q_TRANS_INCREMENT	REAL	Sets the output time increment for the *Flow rate Observation* points.
Q_TRANS_OBS_CELLS_BEGIN	none	Marks the end of an *Transport Observation* point list.
Q_TRANS_OBS_CELLS_BEGIN	none	Marks the end of an *Transport Observation* point list.

5.7.6 Statistics

CMS V4.0 has the option to calculate statistics over the whole model domain for a user-specified time period. This option is accessed using the advanced cards. The starting time, end time, and time interval should be specified in hours with respect to the model start time. The time interval should be larger or equal to the hydrodynamic time step. When activated the global statistics will be output in the same solution file within a subfolder named *stats*.

This option outputs the statistics for hydrodynamics, sediment and salinity transport. If only the statistics for one group is selected then the stats file only include the statistics that belong to those groups.

- **Hydrodynamic statistics:**
    1. Maximum current velocity
    2. Maximum water level
    3. Residual currents (vectors and magnitude)
    4. Hydroperiod
    5. Maximum spatial gradient for water levels
    6. Maximum spatial gradient for current magnitude

- **Sediment Transport and Morphology Change:**
    1. Maximum total load transport rate, m^2/hr
    2. Net total load sediment transport rates, m^2/hr
    3. Average total load sediment transport rates, m^2/hr
    4. Gross total load sediment transport rates, m^2/hr
    5. Positive and negative total load transport rates (in x and y directions), m^2/hr
    6. Maximum spatial gradient of bathymetry

- **Salinity Statistics:**
    1. Mean Salinity

Table 5.7.6. CMS-Flow cards related to output statistics

Card	Arguments	Description	Default value
GLOBAL_STATISTICS	[t0] [tn] [dt]	Calculates global statistics if specified	none
FLOW_STATISTICS	[t0] [tn] [dt]	Calculates flow statistics if specified	none
SEDIMENT_STATISTICS	[t0] [tn] [dt]	Calculates sediment statistics if specified	none
SALINITY_STATISTICS	[t0] [tn] [dt]	Calculates salinity statistics if specified	none

5.7.7 ASCII Output Files

In addition to the XMDF output file, CMS-Flow provides the output two types of ASCII output files:

1. Tecplot snap shot (*.dat), and history files (*.his)
2. SMS Super ASCII files (*.sup, *.xy, *.dat)

The CMS-Flow cards used for outputting these two types of files are described in Table 5.7.7 below.

Table 5.7.7. CMS-Flow cards used to output Tecplot and SMS Super ASCII files.

Card	Arguments	Description	Default value
GLOBAL_TECPLOT_FILES	ON | OFF	Outputs Tecplot ASCII files	OFF
GLOBAL_SUPER_FILES	ON | OFF	Outputs Tecplot ASCII files	OFF

5.8 Steering

Steering refers to the coupling process between CMS-Flow and CMS-Wave. In CMS-Flow versions v3.75 and older (explicit CMS-Flow), the steering was done by the SMS interface. The new inline CMS contains both CMS-Flow and CMS-Wave and performs the coupling process internally. In either, the steering process is similar. First the wave model is run twice, at time zero, until the first steering interval. The wave information is then interpolated on to the flow grid, and the flow model is run from time zero to the first steering interval. Then, the flow information is interpolated on the wave grid, the wave model is run for the second steering interval, and the process is repeated until the simulation is complete.

Before running steering, it is a good idea to test the CMS-Flow and CMS-Wave separately to make sure there are no problems with the grids, or input parameters. Once the CMS-Flow and CMS-Wave models have been setup properly and loaded in SMS, the steering can be initiated.

- **Important Notes**:

    1. For both the SMS steering and inline steering, the CMS-Wave input spectra need to be spaced at regular time intervals and begin at the same time as the CMS-Flow model.
    2. The way variables are interpolated and extrapolated both in space and time are slightly different between the SMS steering and inline steering versions of CMS.
    3. Currently, the inline version of CMS only contains the implicit CMS-Flow solution scheme. Therefore, if the user decides to switch from explicit to implicit solvers, the user must also change the CMS-Flow executable under the SMS preferences.

5.8.1 CMS-Flow Versions 3.75 and Older

In CMS-Flow versions 3.75 and older, the steering process is done by the SMS interface. The interface does all of the variable interpolation and passing of variables from one model to another using communication files. The advantage of this approach is that it keeps the CMS-Flow and CMS-Wave codes separate and makes them easier to maintain and update.

- Start Steering Module to run CMS (with CMS-Flow v3.75)

1. Make sure both the CMS-Flow and CMS-Wave grids are loaded in SMS.

2. Check the CMS-Flow and CMS-Wave file names under the SMS Preferences menu (see Figure 5.8.1). Click on *Edit | Preferences*. Under the *File Locations* tab, in the section called *Model Executables* check the file names for CMS-Flow and CMS-Wave and make sure they are consistent with the latest releases (http://cirp.usace.army.mil/products/index.html CIRP Products).

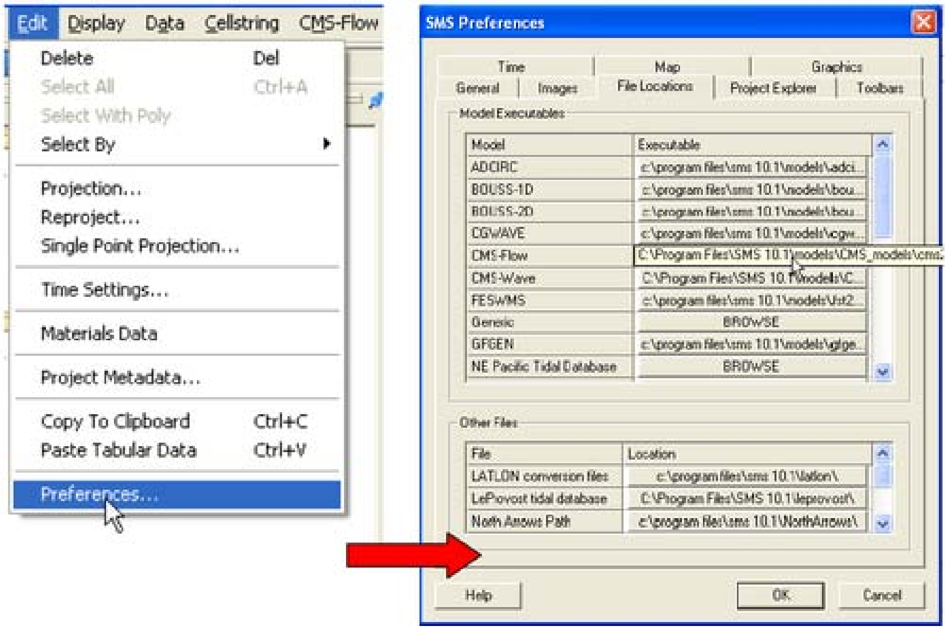

Figure 5.8.1. ***File Location*** **tab within the** ***SMS Preferences*** **window.**

3. Open the SMS Steering Wizard by clicking on the menu *Data* | *Steering Module*. Click on *CMS-Flow <-> CMS-Wave*, click next.

4. Enter the steering interval next the *Time* section. Check the variables which should be passed from one model to the other and select *Start*.

5. A window will appear that says *Are you sure you want CMS-Wave to run every <x> hours*. Make sure the steering interval is correct and click *Yes* to start the steering module or click *No* to exit.

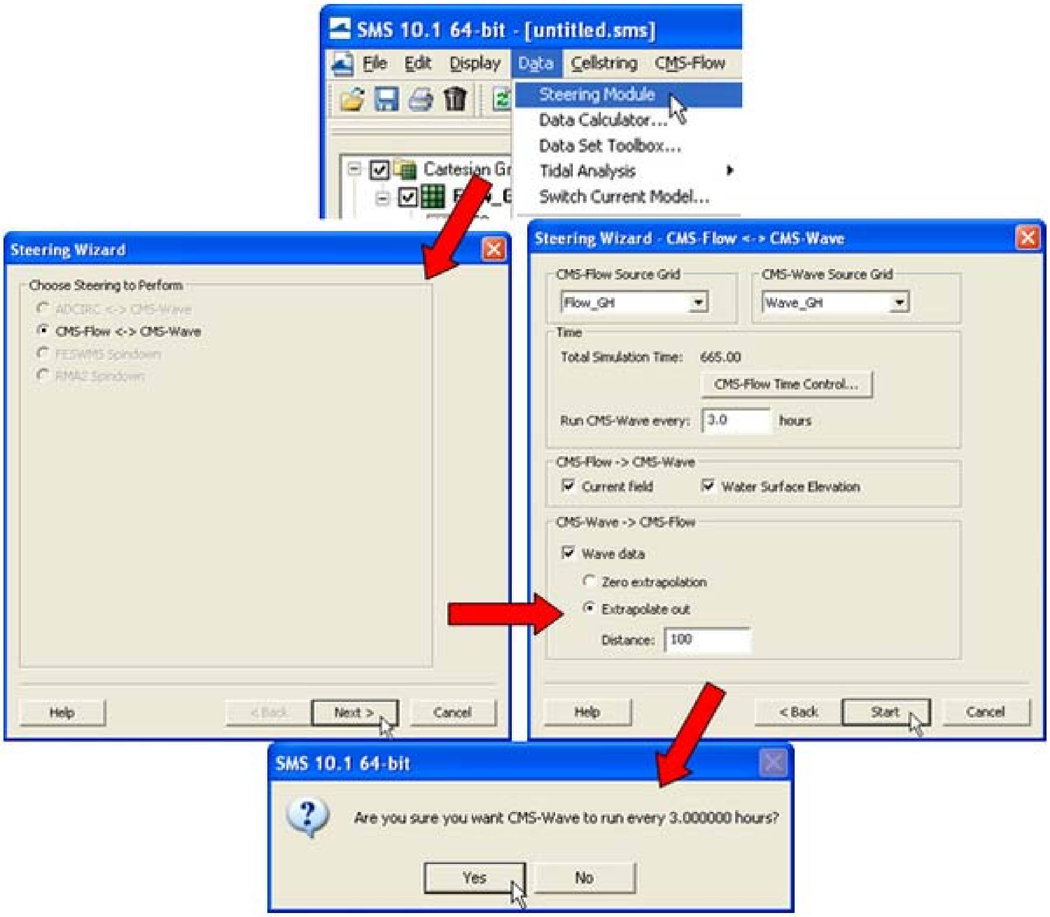

Figure 5.8.2. SMS Steering Module for CMS v3.75 and older.

5.8.2 CMS-Flow Versions 4.0 and Newer

CMS Versions 4.0 and newer the steering process is done internally by the CMS. This means that that both CMS-Flow and CMS-Wave are contained within a single code or executable. Even though CMS-Flow and CMS-Wave use different grids, the two models are in a single code which facilitates the model coupling and speeds up the computation by avoiding communication files, variable allocation and model initialization at every steering interval.

The inline CMS can be launched from the SMS Steering Wizard or as a command line with arguments specifying the input files and steering options. Table 5.8.1 describes the CMS-Flow cards used for the steering process in the inline CMS.

Table 5.8.1. CMS-Flow cards related to steering

Card	Arguments	Default	Range	Description
CMS-WAVE_SIM_FILE	CHARACTER	none	none	File name including path for the CMS-Wave sim file.
WAVE_SIM_FILE	CHARACTER	none	none	Same as CMS-WAVE_SIM_FILE.
STEERING_INTERVAL	REAL	none	none	Sets the recurring hot start output time.
WAVE_WATER_LEVEL	CHARACTER	TIDAL_PLUS_VARIATION	LAST | TIDAL | TIDAL_PLUS_VARIATION	Determines the method used to calculate the water levels passed to the wave model.
FLOW_EXTRAPOLATION_DISTANCE	REAL	Calculated based on grid geometry	none	Determines the extrapolation distance used for flow variables on the wave grid.
WAVE_EXTRAPOLATION_DISTANCE	REAL	Calculated based on grid geometry	none	Determines the extrapolation distance used for wave variables on the flow grid.

- **Interpolation Files**
  When running the inline CMS with flow and waves, the CMS steering module writes out two files named:

  - Intpcoef_flwav.bin
  - Intpcoef_wavfl.bin

  These files contain the interpolation information between the CMS-Flow and CMS-Wave grids. Because calculating the interpolation information can take several minutes, saving this information in files allows the model to quickly read this information and avoid their computation for subsequent runs when using the same CMS-Flow and CMS-Wave grids. When the model is restarted it will automatically detect these files and read them if the grids are the same size. If changes have been made to the grids but the grid size is the same, the steering module will not be able to detect the changes and the interpolation information will be incorrect. Therefore, it is recommended to delete the interpolation files every time a change is made the either the CMS-Flow or CMS-wave grid. In the future, this problem will be avoided by writing a counter to the CMS-Flow and CMS-Wave files every time a change is made to them from the interface.

SMS Steering Module

The inline CMS can be launched from the SMS 11.0 Steering Module in a similar way to previous versions of CMS. The steps for launching the inline CMS (versions 4.0 and higher) are outlined below.

- Start Steering Module to run inline CMS (with CMS-Flow v4.0)

1. Make sure both the CMS-Flow and CMS-Wave grids are loaded in SMS.

2. Check the CMS executable file name under the SMS preferences menu (see Figure 5.8.3). Click on *Edit | Preferences*. Under the *File Locations* tab, in the section called *Model Executables* check the file name for CMS-Flow executable and make sure it is consistent with the latest releases (http://cirp.usace.army.mil/products/index.html CIRP Products).

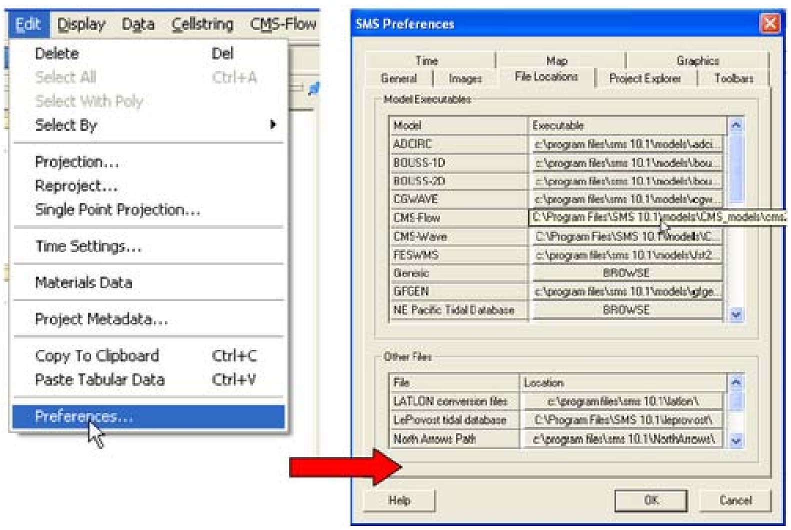

Figure 5.8.3. Changing the CMS-Flow model executable.

3. The steering information must be set in the Advanced Cards click on *CMS-Flow | Model Control…* (Figure 5.8.4). Enter the steering interval and CMS-Wave *.sim file using the cards described in the table above. Note that the full path to the *.sim file must be provided within quotation marks.

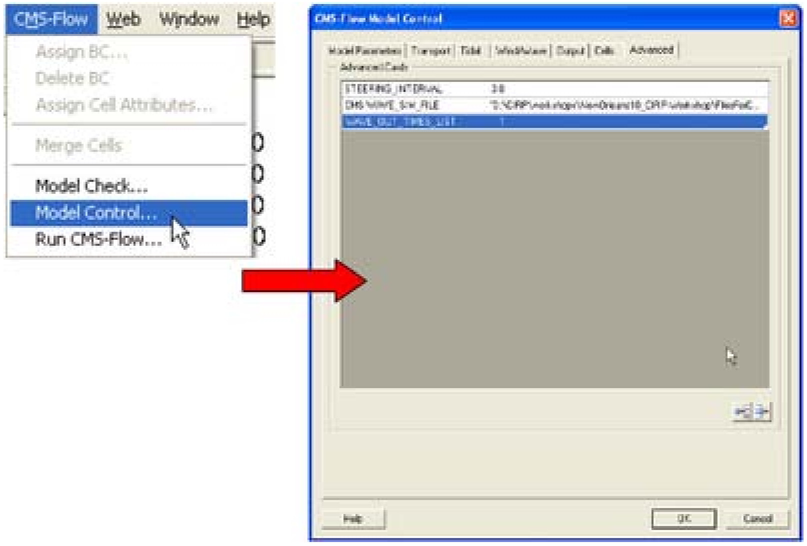

Figure 5.8.4. Setting the CMS steering information.

4. Click on *Data | Steering Module*. In the *SMS Steering Wizard* select the *CMS INLINE* option and *Next*.

5. Enter the *Steering Interval* under the *Time* section and select *Start*.

- **Notes:**
  1. When Running the inline CMS from SMS, it is not necessary to specify the CMS-Wave Sim File or Steering Interval in the advanced cards section.
  2. Steering options besides the CMS-Wave Sim File and Steering Interval such as the extrapolation distances need to be specified in the Advanced Cards.

CMS-FLOW Menu

The second option for launching CMS versions 4.0 and newer is by entering the steering information in the cmcards file and simply running from the SMS CMS-Flow menu. The steps are outlined below.

- Second option to run inline CMS (with CMS-Flow v4.0)

  1. Make sure both the CMS-Flow and CMS-Wave grids are loaded in SMS.

  2. Check the CMS executable file name under the SMS preferences menu (see Figure 5.8.5). Click on *Edit | Preferences*. Under the *File Locations* tab, in the section called *Model Executables* check the file names for CMS-Flow and CMS-Wave and make sure they are consistent with the latest releases (http://cirp.usace.army.mil/products/index.html CIRP Products).

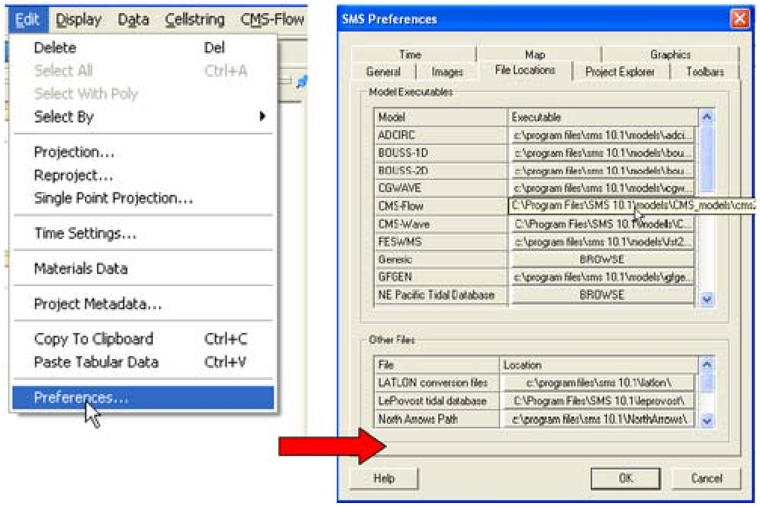

Figure 5.8.5. Changing the CMS-Flow model executable.

3. Start CMS from the CMS-FLOW Menu. Click on *CMS-FLOW | Run*

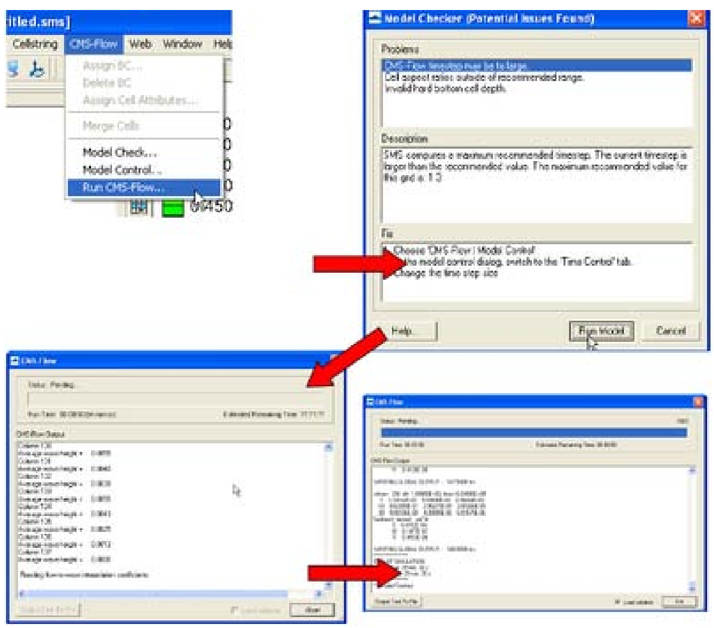

Figure 5.8.6. Example of launching the CMS from the CMS-FLOW menu.

Standalone Program

Standalone refers to the fact that the CMS executable is not part of a larger software package nor does it require a network connection, and support or services of the operating system or other software. If the CMS could only be run from the SMS then it would not be considered a standalone program.

Because the inline CMS does all of the steering internally, it is a standalone program and there is really no need for CMS to be launched from SMS. Running CMS outside of SMS is useful because it allows the user to launch CMS from script files and also to pause the model for checking model results during the model simulation without causing access errors. When running several models, pausing some of them will free up some of the computer to do other tasks such as plotting and checking model results. Running CMS outside of SMS also avoid the extra memory and work requirements from the interface.

- There are two mean approaches for running CMS as a standalone program

  1. From a command prompt.

  2. Double-click on the CMS executable or shortcut to the executable.

  3. Drag-and-drop the CMS input files on the CMS executable or shortcut to the executable.

Notes:

1. For advanced users, it is recommended to put a copy of the CMS executable in the project directory and running the CMS from a command prompt. This keeps a record of the executable used for the project, facilitates making and transferring script files for running multiple project alternatives and keeps the window open after the model has completed or even crashed.
2. To pause the simulation, press the "Pause/Break" button on your keyboard.
3. To stop the model simulation, press and hold the "Ctrl" key and press the "C" key.

- **Running in Command Prompt**

  1. There are two ways of opening a command prompt. The first method for opening a command prompt is to click on Windows *Start | Accessories | Command Prompt* (Figure 5.8.7).

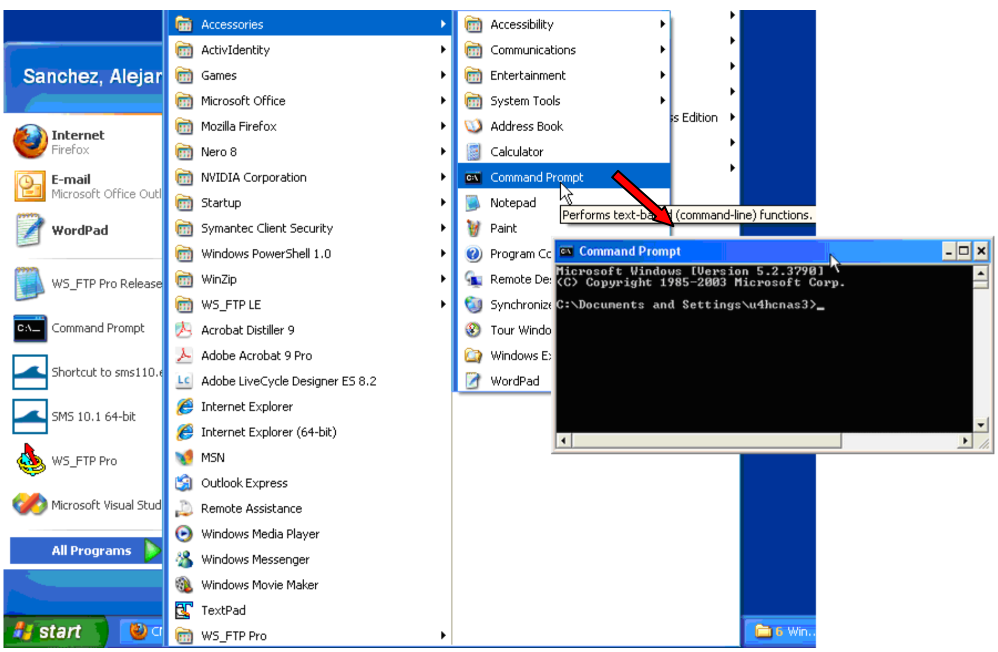

Figure 5.8.7. Example of launching the inline CMS-Flow steering run.

2. The second method for opening a command prompt is to click on Windows *Start* menu then the *Run…* utility. In the *Run* window, enter the command *cmd* and click *OK* (Figure 5.8.8).

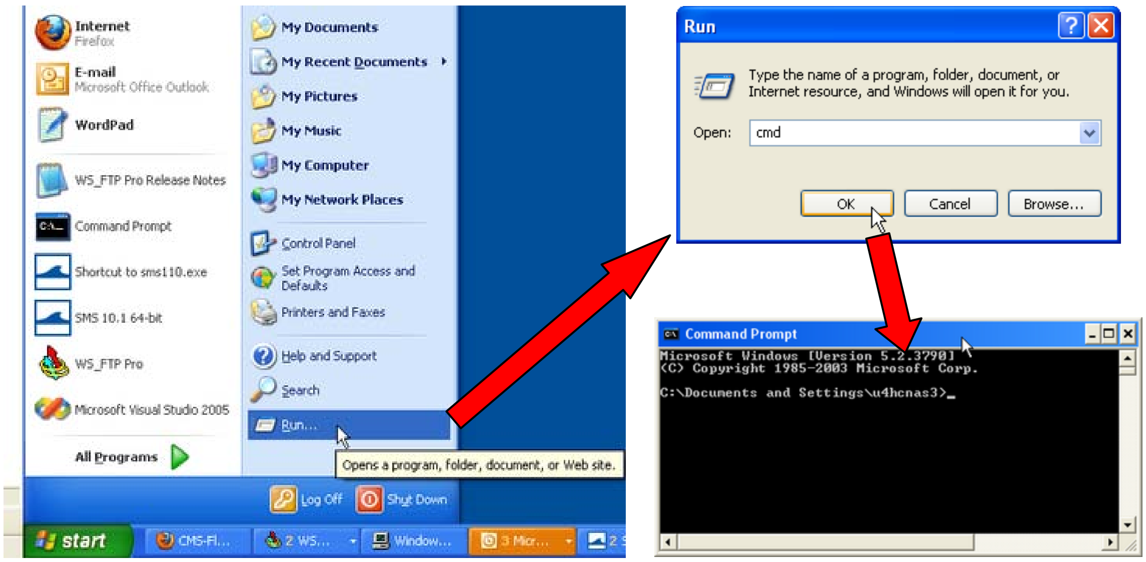

Figure 5.8.8. Example of launching the inline CMS-Flow steering run.

Note: It is recommended to put a shortcut to the Command Prompt on the desktop to more easily open the Command Prompt.

3. Once the command prompt is open the CMS can be launched using one of the following syntax

   >> [cms2d_*exe] [sim or cmcards file] [sim or cmcards file] [steering interval] [wave water level option]

   where the steering interval is in hours and the wave water level option is either:

- 1 – Wave water levels are estimate as the last water levels from the flow model (i.e. WAVE_WATER_LEVEL option equal to LAST)
- 2 – Wave water levels are estimated as the mean tidal water level at the wave time step (i.e. WAVE_WATER_LEVEL option EQUAL TO TIDAL).
- 3 – Wave water levels are estimated as the mean tidal water level at the wave time step plus the water surface variations estimated from the last flow time step (i.e. WAVE_WATER_LEVEL option EQUAL TO TIDAL_PLUS_VARIATION).

Important Notes:

1. The sim and cmcards files must contain the full or relative path with respect to the executable if different from the executable.
2. If no input arguments are specified, than the user will be prompted to manually enter the name of the CMS-Flow and CMS-Wave files and steering information.
3. If the cmcards file is specified the CMS will check for the steering cards. If the sim file is found, than it will run in steering.
4. If no steering interval is specified in the cmcards file or the command line, than a default value of 3.0 hours will be used.
5. If no wave water level option is specified in the cmcards file or the command line, than a default method equal to three.
6. It is possible to create a short-cut to the model executable and simply drag-and-drop the cmcards file and or sim file with the steering options specified in the cmcards file.

Below are some examples:
>> cms2d_v4b43-x32.exe
>> cms2d_v4b43-x32.exe Flow.cmcards
>> cms2d_v4b43-x32.exe Flow.cmcards Wave.sim
>> cms2d_v4b43-x32.exe Wave.sim Flow.cmcards
>> cms2d_v4b43-x32.exe Flow.cmcards Wave.sim 1.0
>> cms2d_v4b43-x32.exe Wave.sim Flow.cmcards 3.0 1

- **Drag-and Drop**

1. To launch CMS as a standalone application using the drag-and-drop method select all of the CMS input files by holding the "Ctrl" key and single-clicking on each input file.

2. Drag all of the input files on the CMS executable or shortcut to the executable by holding the left mouse button.

3. Drop the files by letting go of the left mouse button.

- **Note:** If running steering, make sure all of the steering options are in the cmcards file when using the method.

- **Double Click**

This is one of the easiest ways of running the CMS but also one of the most time consuming because it requires the user to type the name and path (if different from executable) for all of the input files and if necessary the steering options.

1. To run the CMS by using the Double-Click method, Double-click on the executable; Follow the instructions on the screen (Figure 5.8.9).

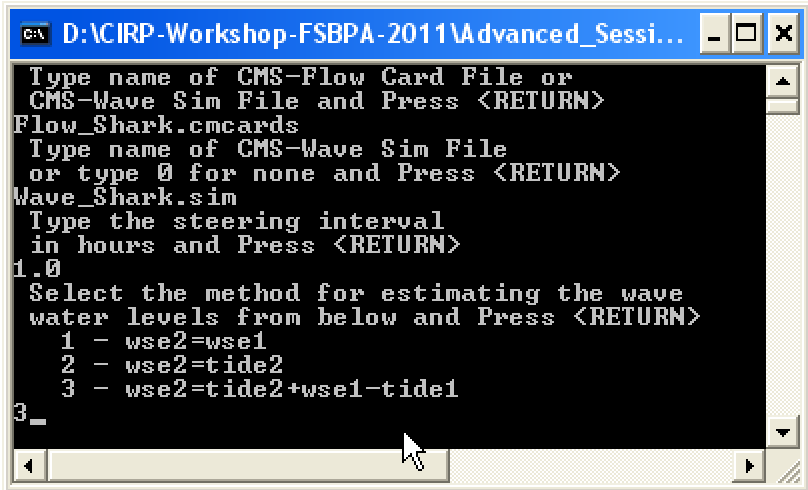

Figure 5.8.9. Example of launching the inline CMS by double-clicking on the executable.

Note: If CMS input files are not in the same folder as the executable, the file names must include the full or relative path with respect to the executable

5.9 Numerical Methods

5.9.1 Solution Scheme

This refers to the temporal discritization of the hydrodynamic, sediment and salinity transport equations. There are two options in CMS: 1. Implicit - First order backward Euler scheme that uses a time step on the order of 5-15 minutes. This is appropriate for cases which can be simulated with large computational time steps such as long term morphology change at inlets. 2. Explicit - First order forward Euler scheme that uses a time step on the order of 0.5-1.0 second. This is appropriate for cases that vary quickly in time such as flooding or barrier island breaching.

Table 5.9.1. CMS-Flow cards related to the surface roller

Card	Arguments	Default	Range	Description
SOLUTION_SCHEME	CHARACTER	EXPLICIT	EXPLICIT | IMPLICIT	Determines the solution scheme used in CMS-Flow.

5.9.2 Solver Options

The four different solvers implemented in the implicit solution scheme are the Gauss-Seidel, Gauss-Seidel with Successive-Over-Relaxation, BICGSTAB, and GMRES. The same solver is applied to flow, sediment and salinity. The default solver is the GMRES. The solver may be changed using the advanced card in table 5.9.2.

Table 5.9.2. CMS-Flow cards related to the solver options

Card	Arguments	Default	Range	Description
MATRIX_SOLVER	CHARACTER	GMRES	GAUSS-SEIDEL | GAUSS-SEIDEL-SOR | BICGSTAB | GMRES	Selects the matrix solver for flow, sediment and salinity.
HYDRO_MAX_ITERATIONS	INTEGER	Function of grid size	>0	Sets the maximum number of iterations for the flow (hydro) solver (outer loop).
PRESSURE_ITERATIONS	INTEGER	Depends on Solver	>0	Sets the number of solver iterations for the pressure equation (inner loop).
VELOCITY_ITERATIONS	INTEGER	Depends on Solver	>0	Sets the number of solver iterations for the velocity or momentum equations (inner loop).
SEDIMENT_MAX_ITERATIONS	integer	20	Maximum number of iterations (outer loop) for the sediment transport	
SALINITY_MAX_ITERATIONS	integer	20	Maximum number of iterations (outer loop) for the salinity transport	

5.9.3 Advection Schemes

As in the case of the implicit solution scheme, the same advection scheme is applied for the flow, sediment and salinity transport equations. There are three choices for advection schemes with upwinding in the implicit model: hybrid, exponential and HLPA. The hybrid scheme is fast but is the most diffusive. The exponential scheme is based on the 1D analytical solution to an advection-diffusion equation and produces very stable results. The HLPA is very stable and non-diffusive, but requires slightly more computational time. For most applications, the exponential scheme is recommended and is set as the default. The advection scheme may be change using the advanced card listed in Table 5.9.3.

Table 5.9.3. CMS-Flow cards related to advection scheme

Card	Arguments	Default	Range	Description
ADVECTION_SCHEME	CHARACTER	EXPONENTIAL	NONE | HYBRID | EXPONENTIAL | HLPA	Sets the advection scheme for flow, sediment and salinity.

5.9.4. Wetting and Drying

Wetting and drying options may be changed using the advanced cards listed in Table 5.9.4.

Table 3.9.4. CMS-Flow cards related to wetting and drying

DRYING_DEPTH	REAL	Calculated based on solution scheme and courant number	none	Sets to the time step for hydrodynamics in seconds.
WATER_PONDING	CHARACTER	OFF	ON | OFF	Turns *On* or *Off* water ponding. If water ponding is *Off*, isolated bodies of water will become dry.
ONE_CELL_WIDE_CHANNELS	CHARACTER	ON	ON | OFF	Limits wetting and drying to areas with at least 3 cells wide. When turned off, the model stability is improved.

5.9.5 Parallelization with OpenMP

Both Intel and AMD processors now are shipping chips with multiple cores/processors (henceforth referred to as "processors") available. CMS-Flow is now configured to make use of these extra processes that are available on newer machines.

Table 5.9.5. CMS-Flow cards related to parallelization with OpenMP

Card	Arguments	Default	Range	Description
NUM_THREADS	INTEGER	1	Determines the number of threads used for parallel processing.	
OPENMP_THREADS	INTEGER	1	Determines the number of threads used for parallel processing.	

5.10 Hot Start

The term Hot start refers to starting a simulation with an initial condition other zero (cold start). Hot starts are used for specifying initial conditions or restarting simulations at intermediate times. The hot start controls are set in the *Flow* tab of the *CMS-Flow Model Control* window.

5.10.1 Hot Start File

The CMS hot start feature CMS lets the user restart simulations that have been stopped due to electric outages, hardware malfunctions, or model crashes. In the case of a model crash the user, may restart the model using larger solver iterations and/or time steps to stabilize the simulation. The user has the option to specify a hot start output time or an interval for outputting a recurring hot start file. Every time the hot start file is written, it overwrites the previous information. The CMS Hot Start file saves information on the water elevation (pressure), and current velocities. If the sediment transport is active, then the water depth, and sediment concentrations are also saved for each size class. The CMS hot start file is a binary XMDF file, has the name Hot_Start.h5 and is saved in the directory of the CMS-Flow files.

Figure 5.10.1 shows the structure of the hot start file. After saving a CMS Hot Start file, it is a good idea to rename the file with a different name before using it as an initial conditions file. This way, the file will not be overwritten in future simulations.

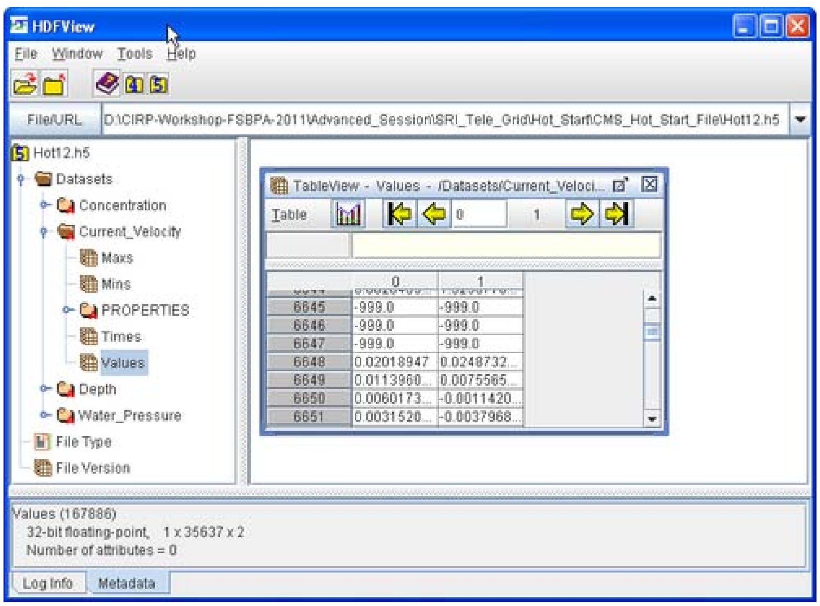

Figure 5.10.1. XMDFView showing the structure of the CMS Hot Start File.

Table 5.10.1. Hot Start CMS-Flow Cards

Card	Arguments	Default	Range	Description
HOT_START_OUTPUT_FILE	CHARACTER	none	none	Julian hour.
HOT_START_TIME	REAL	none	none	Sets the hot start output time.
AUTO_HOT_START_INTERVAL	REAL	none	none	Sets the recurring hot start output time.

5.10.2 Initial Conditions File

There are several situations where it is convenient to specify a user defined initial conditions or hot start file. For example, if the user forgets to setup the model output a hot start file or when running steady state conditions. A hot start file can easily be created and exported by the user from the SMS interface. The model requires water levels, current velocities, concentrations, and water depths. It is important to note that the names and paths of the initial condition datasets are important.

Table 5.10.2. Path and name for initial condition file variables.

Variable	Path and Name
Water surface elevation	Datasets\Water_Elevation
Current velocity	Datasets\Current_Velocity
Sediment concentrations	Datasets\Concentration
Salinity concentrations	Datasets\Salinity

- **Creating a user defined hot start or initial condition file from a CMS-Flow solution file**

  1. Import CMS-Flow grid and solution file. Sample a time step of the solution datasets for use in the initial condition by clicking on *Data | Data Calculator*, and under the *Tools* section select *Sample time steps* (Figure 5.10.2).

  2. Under the *Datasets* section, click on the *Water Elevation*, and export the initial condition datasets to an XMDF file (Figure 5.10.3).

  3. Add the advanced card from Table 5.10.3 to notify the model the initial conditions file name.

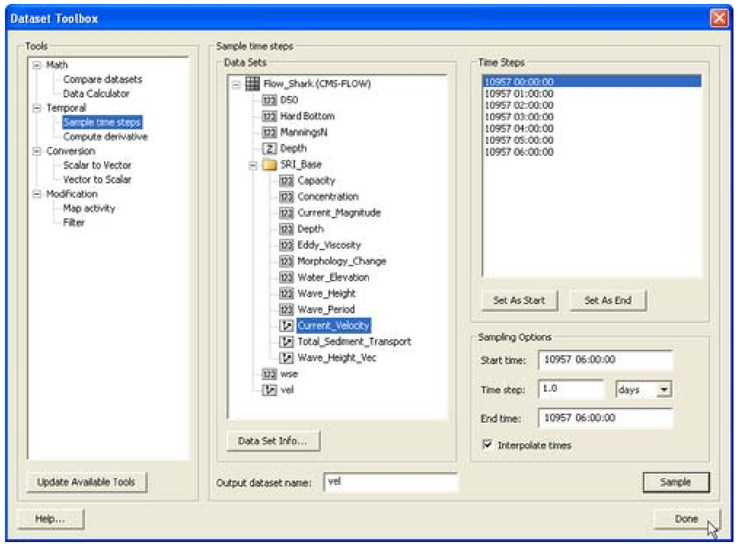

Figure 5.10.2. Dataset Toolbox showing a time step sample of the water elevation and current velocity datasets for use in a hot start (initial condition) file.

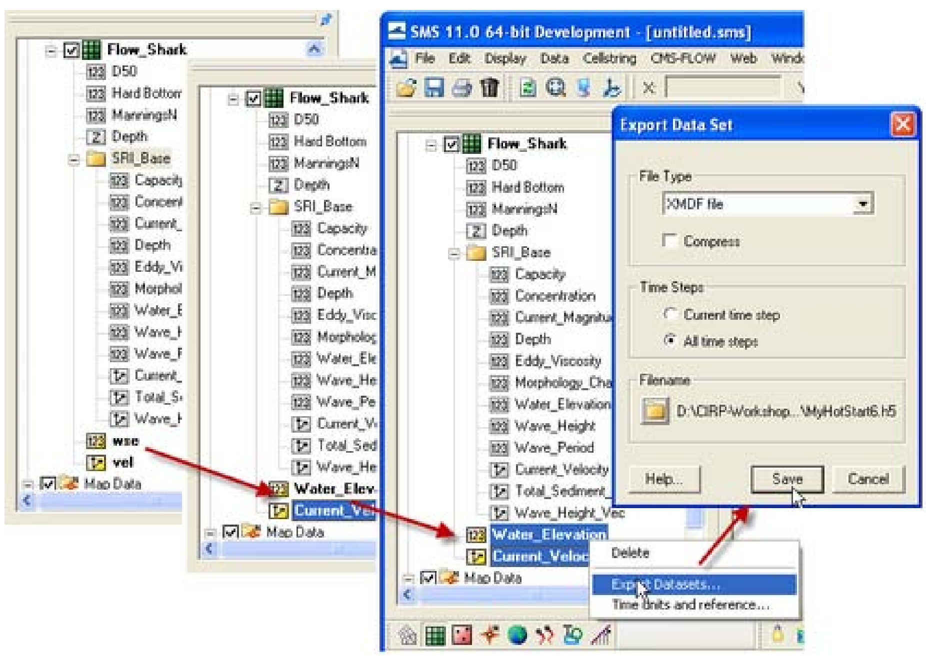

Figure 5.10.3. Dataset Toolbox showing a time step sample of the water elevation and current velocity datasets for use in a hot start (initial condition) file.

Table 5.10.3. CMS-Flow card for specifying the initial condition file.

Card	Arguments	Default	Range	Description
INITIAL_STARTUP_FILE	CHARACTER	none	none	Julian data in YYDDD with YY being last two digits of the year, and DDD the Julian day of the year.

5.11 Scripting

Scipting refers to the automation of running multiple CMS runs with different parameters, without manually having to create and edit each alternative. The scripting process can include the following steps:

1. Setting up alternatives
2. Creating batch file
3. Plotting and analyzing results

Scripting can be done using a variety of software programs. The examples shown here were written in Matlab because it is widely used, easy to read and convenient for plotting and analyzing results.

5.11.1 Setting Up Alternatives

In this example, 4 cases or alternatives (Figure 5.11.1) are set up using the Matlab script below. The script copies the base setup files into subfolders and then modifies specific CMS-Flow cards in the *.cmcards file. The settings for each case are setup using a structure variable with field names corresponding to each CMS-Flow card (e.g. TIME_SERIES_INCREMENT). Separating each case into its own subfolder keeps the input and output separate and also allows for the different cases to be run at the same time.

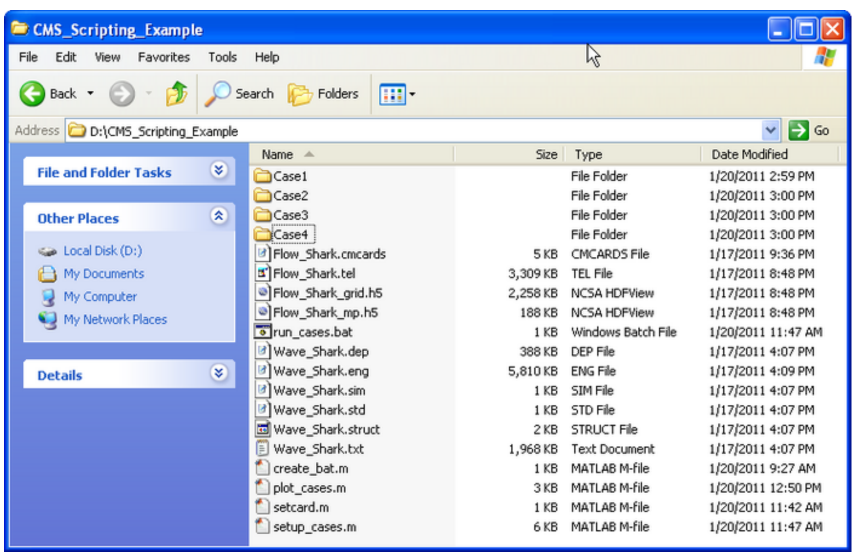

Figure 5.11.1. Example of scripting showing the files used.

```
% Matlab Script: setup_cases.m
clear all
flow = 'Flow_Shark';
wave = 'Wave_Shark';
ncases = 4; %Number of cases or alternatives
r(1).MANNINGS_N_DATASET = '"Manning_Alt1.h5"
"Flow_Shark/Datasets/ManningsN"';
r(1).WAVE_CURRENT_MEAN_STRESS = 'W09';
r(1).TIME_SERIES_INCREMENT = 1800;
r(2).MANNINGS_N_DATASET = '"Manning_Alt1.h5"
"Flow_Shark/Datasets/ManningsN"';
r(2).WAVE_CURRENT_MEAN_STRESS = 'DATA2';
r(2).TIME_SERIES_INCREMENT = 900;
r(3).MANNINGS_N_DATASET = '"Manning_Alt2.h5"
"Flow_Shark/Datasets/ManningsN"';
r(3).WAVE_CURRENT_MEAN_STRESS = 'W09';
r(3).TIME_SERIES_INCREMENT = 900;
r(4).MANNINGS_N_DATASET = '"Manning_Alt2.h5"
"Flow_Shark/Datasets/ManningsN"';
r(4).WAVE_CURRENT_MEAN_STRESS = 'DATA2';
r(4).TIME_SERIES_INCREMENT = 600;
for i=1:ncases
  d = ['Case',int2str(i)];
  if ~exist(d,'dir')
    mkdir(d)
  end
  copyfile([wave,'.*'],d)
  copyfile([flow,'.*'],d)
  copyfile([flow,'_mp.h5'],d);
  copyfile([flow,'_grid.h5'],d)
  cards = fieldnames(r(i));
  file = ['.\',d,'\Flow_Shark.cmcards'];
  fork=1:length(cards)
    setcard(file,cards{k},r(i).(cards{k}));
  end
end
return
```

The script above requires the subroutine below.

```
function setcard(cmcardsfile,card,value)
% setcard(file,card,value)
% Overwrites or appends a CMS-Flow card
% in the *.cmcards file
copyfile(cmcardsfile,'temp')
fid=fopen('temp','r');
fid2=fopen(cmcardsfile,'w');
nc=length(card);
ok = false(1);
if ~ischar(value)
  value = num2str(value);
```

```
end
while 1
  tline = fgets(fid);
  if ~ischar(tline), break, end
  if strncmp(card,tline,nc)
    fprintf(fid2,'%s       %s %s' ,card,value,tline(end));
    ok = true(1);
    continue
  end
  nline = length(tline);
  if (~ok && strcmp(tline(1:min(nline,14)),'END_PARAMETERS'))
    fprintf(fid2,'%s       %s %s',card,value,tline(end));
    fprintf(fid2,'%s' ,tline);
    break
  end
  fprintf(fid2,'%s' ,tline);
end
fclose(fid);
fclose(fid2);
delete('temp')
return
```

5.11.2 Batch File

Although it is possible to launch CMS from Matlab a batch file is preferable to use a batch file because it allows running all of the cases without opening Matlab.

```
% Matlab Script: create_bat.m
cmsexe = 'cms2d_v4b42_x64p.exe'; %CMS-Flow executable
batfile = 'run_cases.bat'; %Output batch file
fid = fopen(batfile,'w');
for i=1:ncases
  cmcards = ['.\Case',int2str(i),'\',flow,'.cmcards']; %CMS-Flow
cmcards file
  fprintf(fid,'START %s %s %s',cmsexe,cmcards,char(10));
end
fclose(fid);
return
```

The following text shows what the resulting batch file (*.bat) looks like

```
START cms2d_v4b42_x64p.exe .\Case1\Flow_Shark.cmcards
START cms2d_v4b42_x64p.exe .\Case2\Flow_Shark.cmcards
START cms2d_v4b42_x64p.exe .\Case3\Flow_Shark.cmcards
START cms2d_v4b42_x64p.exe .\Case4\Flow_Shark.cmcards
```

To run the batch file, simply double click on the file and each case will launch separately in its own MS-DOS window.

5.11.3 Plotting

The following example reads the Observation Point time series output file (*_eta.txt) and plots the 3rd column corresponding to the second observation point.

```
% Matlab Script: plot_cases.m
close all
eta = cell(ncases,1);
for  i=1:ncases
  etafile = ['.\Case' ,int2str(i),'\',flow,'_eta.txt' ]; %Water
elevation
  eta{i} = load(etafile);
end
figure
hold on
for i=1:ncases
  h = plot(eta{i}(:,1),eta{i}(:,3),'-');  %3 is the index is the
observation point index
end
ylabel('Water elevation, m')
xlabel('Elapsed Time, hr')
return
```

References

- Hsu, S.A. (1988) Coastal meteorology. Academic Press, San Diego, CA.
- Shore Protection Manual (1984) 4th ed., 2nd Vol, U.S. Army Engineer Waterways Experiment Station, U.S. Government Printing Office, Washington, DC.
- Folk, R.L., and Ward, W.C. (1957). "Brazos River bar: a study in the significance of grain size parameters", Journal of Sedimentary Petrology, 27, 3–26.