# *SURFACE WATER MODELING SYSTEM*

# ***CMS-Flow2D***

## 1 Introduction

This lesson will teach you how to prepare a grid and run a solution using *CMS-FLOW2D*. The files used by this simulation are referenced through a ".sms" file. The file "*Ideal.sms*" contains a link to the bathymetry file "*Ideal_scatter.h5*" and "*Ideal.map*" files. These files can be found in the Data Files Folder for this tutorial. To open the file:

1. Select *File* | *Open*.

2. Find and highlight the file *Ideal.sms*. If you still have data open from a previous tutorial, you will be asked if you want to delete existing data. If this happens, click the *Yes* button.

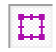
## 2 Creating a Grid

The data provided for you to start this tutorial includes the bathymetry and an arc that defines the shoreline. The bathymetry is in the form of a scattered data set (or triangulated irregular network or TIN) and shoreline is in a coverage named "Ideal_Inlet". Now we want to create a grid that will be used for numerical computations. To create the grid:

1. Click on the "Ideal_Inlet" to make sure you are working in the *Map Module* and the coverage is active.

2. Select the *Select Arc*  tool and click on one of the shoreline arcs. Arrows will appear pointing away from the arc at each end. The blue arrows indicate the side of the arc that will be classified as water. The brown arrows indicate the side that will be classified as land or inactive.

3. Select the *Create 2-D Grid Frame*  tool and define the extents of the computational domain. This is done with three clicks. The first click defines a corner of the domain (usually the origin). Click near the loser left corner of the data for this point. The second click defines the first direction of the grid and extent of the grid in that direction. Click near the lower right of the data for this point. The third click defines the extent of the domain perpendicular to the first direction. Click anywhere near the top of the domain to define this distance. The thick purple line just inside the data extents in Figure 1 illustrates a grid frame.

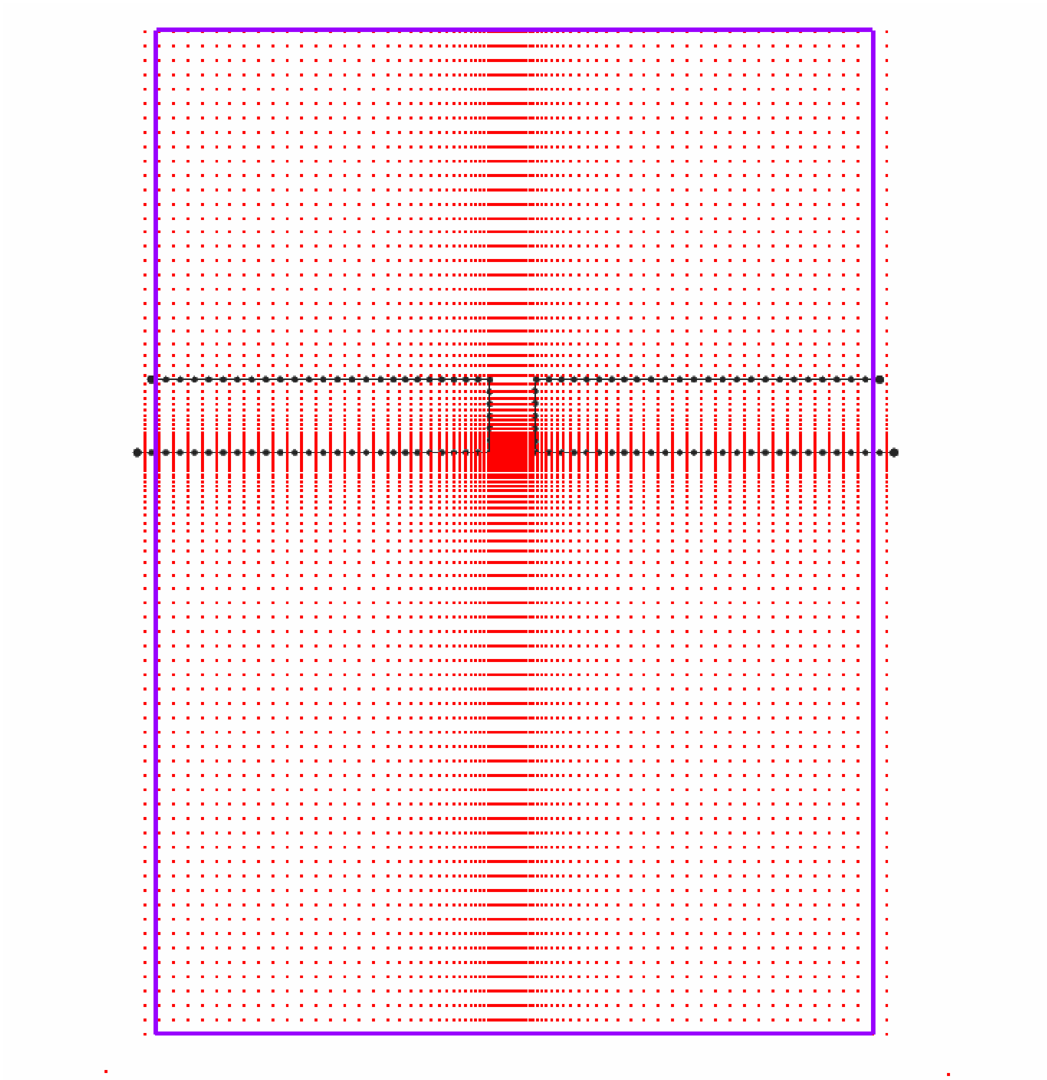


*Figure 1 The scatter set and coverage contained in the file Ideal.sms.*

4. Right click the "Ideal_Inlet" and select *Convert* | *Map* -> *2-D Grid*. The *Map* -> *2D Grid* dialog will appear. This dialog allows you to modify the extents of the grid frame explicitly and set the desired grid attributes. The exact values for the position and size of a grid do not generally matter in a real world simulation. However, for consistency in this tutorial, you should change the origin and

orientation, specified in the upper portion of the dialog to be: origin = (-3030, 300) and I/J size = (1500, 2100). The angle should be zero.

5. In the center section of the dialog select the *I Cell Options* to use "Number of cells" and enter a value of 60 (this is the number of columns). Also set the *J Cell Options* to use "Number of cells" and enter a value of 75 (this is the number of rows). Notice that the cell size is computed and reports that the cells will be 25 x 28 meters.

6. In the Depth Options section, make sure the "Source" is set to "Scatter Set". Press *Select* button and make sure that "depth" data set is highlighted. This defines the source for depth data for the model. (Note that CMS-Flow uses depths (positive measured downward). If a survey is done relative to a sea level datum and the data is measured as positive upwards (elevation), the datum would need to be switched or a depth data set computed using the data calculator.)

7. Click *OK* to exit the dialogs.

SMS creates a Cartesian grid in the grid frame defined and adds it to the project explorer. You will see an object named "Ideal_InletGrid" in the "Cartesian Grid Data" entry of the data tree. SMS has defined the cells, assigned depth values to each cell and created cell strings around the boundaries. Cells on the "land" side of the arcs have been assigned to be inactive (or land).

# 3 Defining Boundary Conditions

## 3.1 General Parameters

Now we have a grid. You have the option of explicitly specifying the status of cells in the grid as land (inactive) or water. To illustrate this:

1. Click on the "Ideal_InletGrid" to make it active and switch to the Cartesian Grid Module.

2. Select the *Select Grid Cell*  tool.

3. Select one or more cells by clicking on them or dragging a box around them.

4. Select CMS-Flow| Assign Cell Attributes. The options for cell type appear in the *CMS-FLOW Cell Attributes* dialog. Since we already have the desired options, click OK to exit the dialog.

## 3.2 Defining Boundary Conditions

For this tutorial, the lower portion of the domain represents the ocean side of a barrier island. The upper portion represents a bay. The land cells represent the barrier island(s).

We will define boundary conditions along the ocean boundary of the grid with a cellstring. This boundary will have a boundary condition that resembles a tidal cycle.

To assign the boundary condition:

1. Choose the *Select Cell String* tool and click anywhere on the cellstring around the ocean boundary (lower portion of the grid).

2. Select *CMS-Flow | Assign BC*.

3. Select *WSE*-forcing and click on the *file button* to browse for a file. Select "Ideal.wl" and click *open*. SMS loads a curve simulating a fluctuation tide. To view the curve, click on it. An *XY Series Editor* appears. This dialog shows the values in the curve on the left and plots the curve on the right. In this case, the time series spans over 52 days, so the curve is hard to see. You can use your mouse to select a sub portion of the plot to zoom in. You must select this region by clicking in the white space above or below the plot and dragging a box around the area you wish to zoom in to.

4. Click *OK* to close both dialogs.

*CMS-Flow2D* supports several other types of boundary conditions that will not be illustrated in this tutorial.

## 3.3 Defining the Model Control

We need to set the parameters for running *CMS-Flow2D*. This is done in the M*odel Control* dialog.

1. Select CMS-Flow | Model Control

2. In the *Flow* tab, choose the starting date and time to Feb 15, 2005 at 10 am.

    - Put the *Simulation duration* at 12 hours. (Note: normal applications span much longer time ranges. This short span is used for demonstration purposes.)

    - Put the *Ramp duration* at 4.8 hours (0.2 days). (Note: normally this would be around two tidal cycles.)

    - For *Hydrodynamic time step* enter 1 second.

    - Set the *Solution Scheme* to "Explicit".

3. The *Sediment* and *Salinity* tabs allow you to turn on the sediment transport and salinity diffusion for a simulation. We will not be using these options in this tutorial, so skip these tabs.

4. The *Tide* tab allows the definition of the tidal constituents that will be applied to boundary locations specified as tidal. This tutorial does not use this boundary condition so skip this tab.

5. The *Wind/Wave* tab allows the definition of the winds and wave options of CMS-Flow. This tutorial does not use these options so skip this tab.

6. In the *Output* tab make sure the “Output Times 1” option is selected. Under Output Times, set the start time to 0, the increment to .25, and the end time to 50. Under *Output Datasets* select “Output Times 1” for *Water surface elevation* and for *Velocity*. This tells CMS-Flow to save water levels and velocities at each cell every 15 minutes. The other output options are not enabled for this simulation.

7. The In the *Cells* tab change the *Output interval* to 15 seconds for both “Time Series Observation Output” and for “Flow Rate Observation Output”. This interval tells CMS the frequency for saving data at specified observation cells of these types. We will define observation cells in the next section. In the “Transport Observation Output” section, turn off the *X* and *Y* directions.

8. In the *Input* tab, set the *Bottom Friction Dataset* to “Mannings N” and click *Create Dataset*. The *Mannings N* dialog will come up. Click OK.

9. Click OK to exit the Model Control dialog.

### 3.4 Defining the Observation Cell

We need to set the observation point on the grid. This is done in the Cartesian Grid module.

1. Select the *Select Grid Cell*  tool.

2. Click on a cell within the inlet.

3. Select CMS-Flow | Assign Cell Attributes…

4. Select *Observation* and turn on *Time series output* and *Flow rate output*. Make sure *Transport output* is turned off.

5. For *Identifier* enter “Point 1”. This would correspond to the name of the gage or another identifying name for a point in the domain.

6. Click OK.

## 4 Saving the Simulation

1. Select *File* | *Save As* and make sure the *Save as type* is set to *Project Files*.

2. Enter *Ideal_out* for the *File name* and press *Save*.

# 5 Using CMS-Flow2D

CMS-Flow2D can be launched from inside SMS. To do this:

1. Select *CMS-Flow | Run CMS-Flow*.

2. *SMS* saves the location of the *CMS-Flow2D* executable as a preference. If this preference is defined, the model will launch. If the preference is undefined, *SMS* shows a message that the executable is not found, click the *File Browser* button to find the *CMS-Flow2D* executable and Click the *OK* button to run the model.

(Note: When before *SMS* launches the model, it performs a quick model quality check. If any problems are detected a message box will be displayed for the user to respond to.)

- When *CMS-Flow2D* finishes running, click the *Exit* button. The run has created a new file in the same directory as the project. The new file is named M2D_Ideal__default coverageGrid_sol.h5

This file can be opened using the *File | Open* command. As it is read, data sets are added to the *Project Explorer* under the grid item. To view the solutions:

1. Go to *Display | DisplayOptions*.

2. Select *Cartesian Grid* and turn on *Contours* and *Vectors*.

3. Switch to the *Contours* tab and change the *Contour Method* to “Color Fill”.

4. Click OK.

5. Click through the timesteps to observe the changes.

# 6 Conclusion

This concludes the *CMS-Flow2D Analysis* tutorial. You may continue to experiment with the *SMS* interface or you may quit the program.