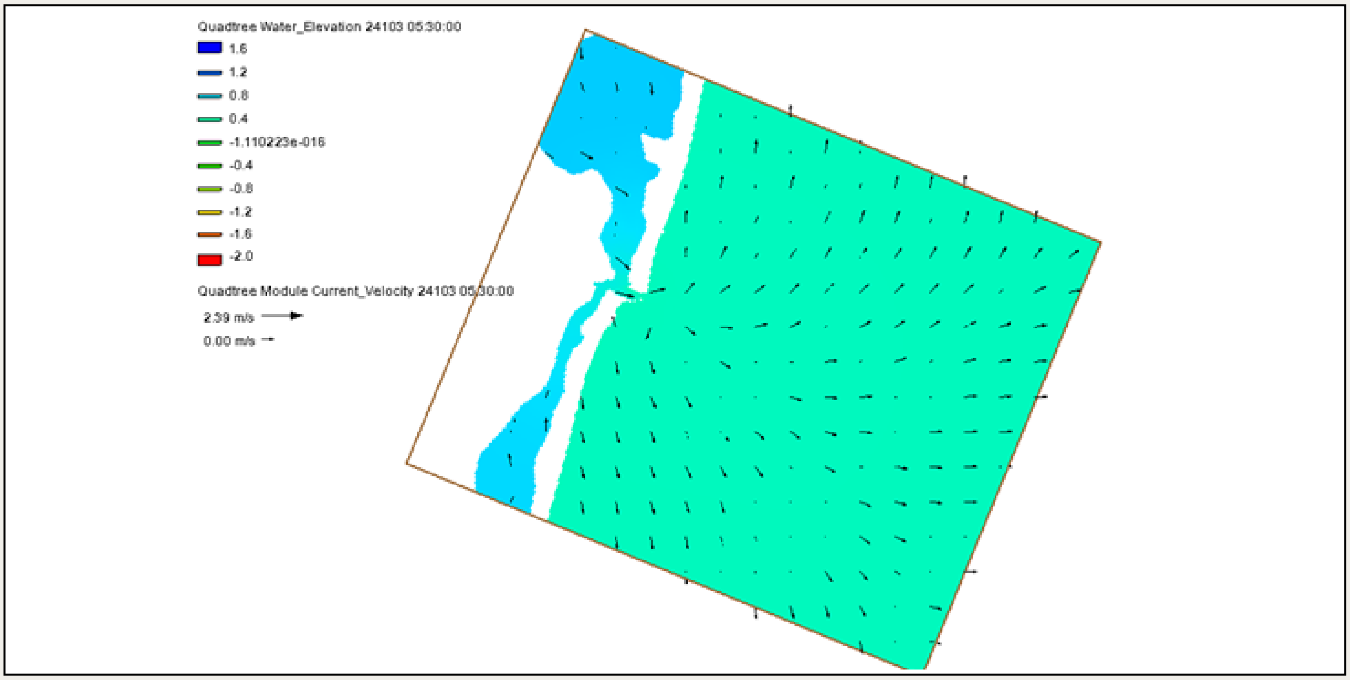
SMS 14.0 Tutorial

CMS-Flow Simulation

Creating a CMS-Flow Simulation



Objectives

This tutorial demonstrates how to create and run a CMS-Flow simulation.

Prerequisite Tutorials

- Overview
- Quadtree Components

Required Components

SMS Core

- CMS-Flow Model & Interface

Time

- 5–10 minutes

1 Introduction

This tutorial demonstrates creating and running a basic CMS-Flow simulation in SMS. This includes:

- Creating the simulation and adding the components to the simulation.
- Specifying model parameters.
- Executing the CMS simulation.
- Reviewing the simulation results.

Previous tutorials illustrated the creation of the quadtree and CMS-Flow components.

The data used in this tutorial is from the Ocean City inlet and the surrounding coastline in the state of Maryland in the United States. The projection for the project has been preset.

2 Getting Started

Start by importing a project file containing the data for this CMS-Flow simulation:

1. Launch or restart SMS.
2. Select *File* | **Open…** to bring up the *Open* dialog.
3. Select “Project Files (*.sms)” from the *Files of type* drop-down.
4. Browse to *data files* folder and select “ocean_city_components.sms”.
5. Click **Open** to import the project and exit the *Open* dialog.

The project should appear similar to Figure 1. The project includes raster data, predefined map coverages, and UGrids. The existing components will be used with the CMS-Flow simulation.

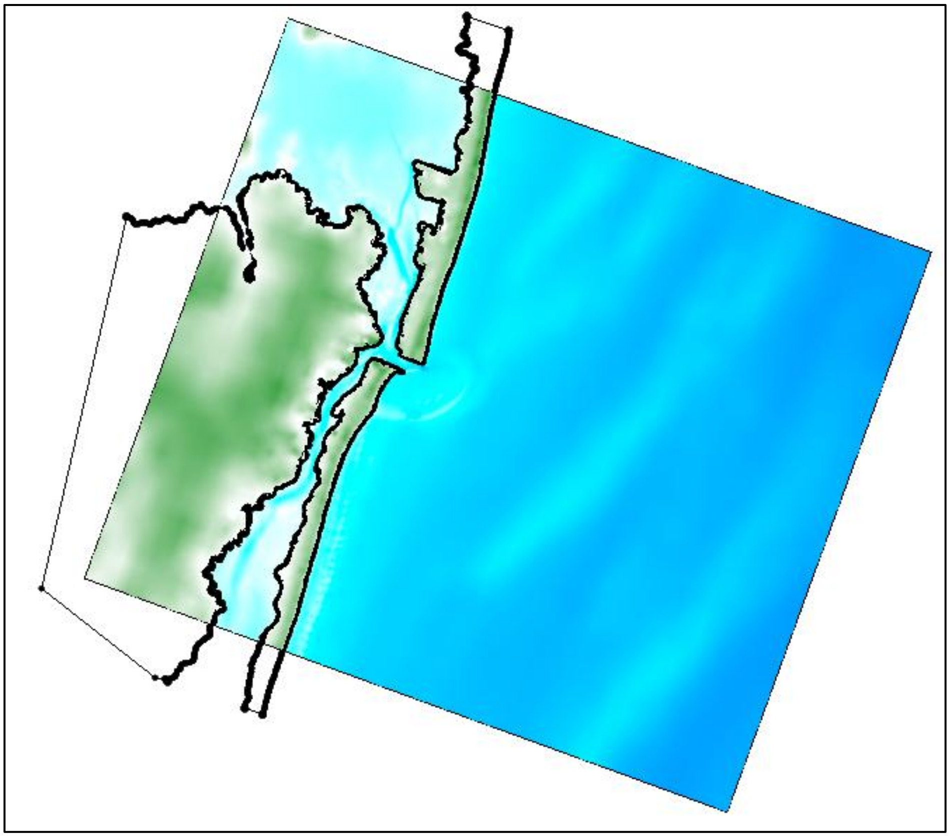


Figure 1 Ocean City inlet

3 Setting Up the Simulation

The CMS-Flow interface makes use of simulations. Multiple simulations can be included in a project. Set up a CMS-Flow simulation by doing the following:

1. Right-click in a blank area in the Project Explorer and select *New Simulation* | **CMS-Flow**.

This creates in the Project Explorer a new " Sim" under a " CMS-Flow Simulations" folder under " Simulation Data".

2. Right-click on " Sim" and select **Rename**.
3. Enter "Ocean City Tides" and press *Enter* to set the new name.
4. Right-click " Refinement Zones Grid" and select *Apply to* | **CMS-Flow Simulations → Ocean City**.
5. Repeat step 4 for " Tidal BC", " Activity", and " Save Points".
6. **Frame** the project.

The Project Explorer should now appear similar to Figure 2.

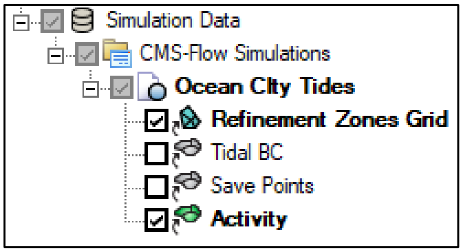


Figure 2 Components linked to the simulation in the Project Explorer

4 Defining the Model Control

Set the parameters for running CMS-Flow in the *CMS-Flow Model Control* dialog by doing the following:

1. Right-click on " Ocean City Tides" and select **Model Control…** to bring up the *CMS-Flow Model Control* dialog.

This tutorial will step through the tabs in this dialog needed for a basic CMS-Flow simulation.

4.1 General Tab

The *General* tab includes the date, time range, and general operating parameters.

1. Select the *General* tab if it is not already active.
2. For the *Start date/time,* enter "9/1/2020 1:00 PM".
3. For the *Simulation duration,* enter "48.0" and select "hours" from the drop-down.
4. For the *Ramp duration,* enter "24.0" and select "hours" from the drop-down.

Full tidal runs often require a full lunar cycle of 28 days or at least 14 days. This run is shorter to allow for shorter run times.

5. Make sure *Second order skewness correction* is turned on.
6. In the *Hot start* section, make sure *Initial conditions file* is turned off.

This option instructs CMS-Flow to start from cold start conditions.

7. Set the *Single hot start time to write out* to "48.0" and "hours" to tell CMS-Flow to write a hot start file at the end of the simulation.
8. Turn off the *Write recurring hot start file* to tell CMS-Flow to not save additional hot start files.
9. In the *Solution scheme* section, select "Implicit" from the first drop-down.
10. For the *Matrix solver,* select "GMRES" from the drop-down.
11. In the *Threads* section, for the *Number of threads* enter "4".

4.2 Flow Tab

The *Flow tab* includes parameters related to the hydrodynamic computations.

1. Select the *Flow* tab to make it active.

2. In the *Parameters* section, for the *Hydrodynamic time step,* enter “600” and select “seconds” from the drop-down.

This defines the maximum time step to be 600 seconds (10 minutes). CMS-Flow will reduce this if needed to maintain stability.

3. Scroll down to the *Bottom and wall friction* section, for the for *Bottom roughness type* select “Mannings N” from the drop-down.
4. For *Bottom Roughness*, click **Dataset** to bring up the *Select bottom roughness dataset* dialog.
5. Select “n = 0.025” from the dataset list.
6. Click **OK** to close the *Select bottom roughness dataset* dialog.

4.3 Output Tab

For this simulation, no sediment transport, salinity, wave, or wind options will be included. These functions are off (or set to “none”) by default, so no changes will be made on the *Sediment Transport*, *Salinity/Temperature*, *Wave*, and *Wind* tabs. Feel free to examine each of these tabs to review the default settings. Additional tutorials will describe using these options.

For now, proceed to the *Output* tab which includes parameters related to the desired output from the numeric engine.

1. Select the *Output* tab to make it active.
2. For the *Simulation label,* enter “Ocean City”.
3. In the *Output time lists* section, under *List 1*:
   - For the *Start time (hrs)* column, enter “24”.
   - For the *Increments (hrs)* column, enter “1”.
   - For the *End time (hrs)* column, enter “720”. This is longer than the simulation, but that is fine.
4. Turn on the option to output *Current magnitude (m/s)*.
5. Scroll down to the *Output options* section and change the *Solution/statistical output options* setting to “XMDF binary output”.
6. Turn on the *Use single XMDF solution file (_sol.h5)* option.
7. Click **OK** to close the *CMS-Flow Model Control* dialog.

The other output options are related to the features that are not illustrated in this tutorial

5 Saving the Simulation

It is recommended to save the project prior to running CMS-Flow.

1. Select *File* | **Save As*…*** to bring up the *Save As* dialog.
2. Select “Project Files (*.sms)” from the *Save as type* drop-down.
3. Enter “ocean_city_sim.sms” as the *File name.*
4. Click **Save** to save the project under the new name and close the *Save As* dialog.

6 Running CMS-Flow

CMS-Flow can be launched from inside SMS by doing the following:

1. Right-click on " Ocean City Tides" and select **Save Project, Simulation and Run...** to bring up the *Simulation Run Queue* dialog.

If any problems are found in the model quality check, the problems will appear in the text field at the bottom of the dialog. The steps needed to correct the problems will also be displayed.

SMS saves the location of the CMS-Flow 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, and the location of the executable must be specified.

2. If this happens, click the **File Browser** button to browse and find the CMS-Flow executable, then click the **OK** button to run the model.
3. Click **Yes** when advised the "Activity" coverage will be renumbered.

The "Ocean City Tides" simulation should be at the top of the list of simulations. There should be only one simulation in the list at this time.

Text messages in the *Command line* portion echo the notices from the CMS-Flow engine as it runs. Notices are provided for a number of events including:

- Stability status and time step size for each time step. The time step may be reduced automatically to maintain stability.
- When global out is saved.
- When hot start files are written.
- Elapsed clock time with an estimated completion time on a periodical basis.

Continue reviewing the *Simulation Run Queue*.

4. Select the *Residual Plot* tab to view the numerical stability of the run.

These curves should maintain small numbers. If the residual values get too large, CMS-Flow automatically reduces the time step size.

Depending on computer speed, this two day run could take as little as 5 minutes or as much as 20 minutes.

7 The CMS-Flow Solution

Once the simulation is completed the progress bar in the model run queue turns green and the *Abort* button changes to read *Load*. It can be useful to review the command line output from CMS before loading the solution. This output includes any diagnostics from the engine as well as the total CPU time required to run the simulation.

7.1 Loading the Solution

To load the solution:

1. Click on **Load Solution** to load the solution datasets.

The solution sets will appear in the Project Explorer in two places. The times that appear in the time window do not make sense because the reference time in SMS does not match the simulation time.

2. Right-click on *Time steps* and select **Time Settings** to bring up the *Time Settings* dialog.
3. Set the *Zero time (reference time)* to “9/1/2020 1:00:00 PM” to match the simulation.
4. Click **OK** to close the *Time Settings* dialog.

7.2 Setting Display Options

The principal outputs of CMS-Flow include water levels and current velocities at each cell. We have been viewing contour displays of ground elevation. Similar display techniques will be used for water levels. However, we need to set the options to display the currents (vectors). To do this:

1. Select “ Water_Elevation” to make it active. The current dataset is the only vector dataset so it will be active by default.
2. Expand the “ Display Themes” folder in the Project Explorer.
3. Expand the “ Contours” folder and click on the “ Depth” theme to change the contour options to shades of blue.
4. Right-click on the “ Refinement Zones Grid” and select **Display Options…** to open the *Display Options* dialog.
5. Turn on *Vectors*.
6. Next to the *Vectors* options, click the **Options…** button to bring up the *Vector Display Options* dialog.
7. In the *Arrow Options* section, turn off *3D Arrows*.
8. Set the *Shaft length* to “Define min. and max. length”.
9. Set the *Maximum* to “50” (pixels).
10. Turn on the *Arrows follow flow path* option.
11. In the *Color Range* section, click on the **Options…** button to bring up the *Choose color ramp* dialog.
12. Expand the *Colorcet* folder.
13. Expand the *Diverging* folders.
14. Select the “CET_D3” color ramp to display vector arrows in a range of colors different from the contours.
15. Click the **OK** to close the *Choose color ramp* dialog.
16. In the *Vector Display Placement and Filter* section, select “on a grid” from the *Display* drop-down.
17. Enter “30” (pixels) for both *X spacing* and *Y spacing*.
18. Set the *Origin* to “Relative to max elevation”.
19. Set the *Offset* to “5.0” to make sure the vectors are above the contours.
20. Click **OK** to close the *Vector Display Options* dialog.
21. Click **OK** to close the *Display Options* dialog.
22. **Zoom** into the inlet and select the “1 03:00:00” time step.

The screen should be similar to the figure shown in Figure 3

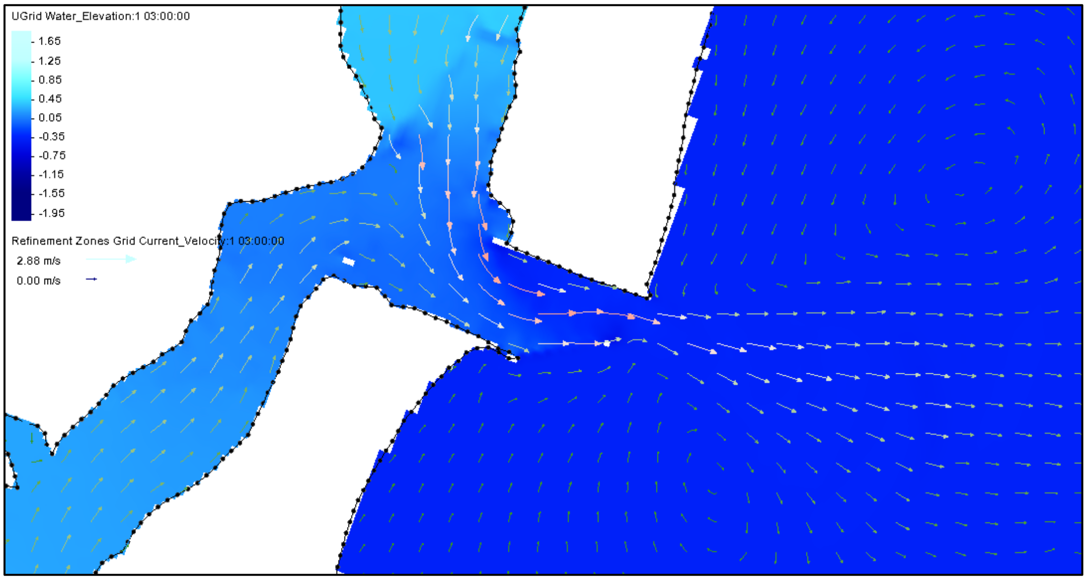


Figure 3 Time step 1 03:00:00

7.3 Stepping through Time Steps

Click through different times in the *Time steps* window to view how the flow patterns and water levels change through the 2nd day of the simulation. Notice the eddy patterns of velocity outside of the inlet as well as the water level changes through the inlet.

8 Conclusion

In addition to the datasets reviewed in the previous section, CMS also creates time series output for each save point. These are included in text files in the simulation folder. This includes water surface elevation data in the "_eta.sp" file and velocity data in the "_uv.sp" file. There are also times series files for unit flow rates and total depth.

This concludes the "CMS-Flow Simulation" introductory tutorial. Feel free to continue experimenting with the SMS interface or exit the program.

