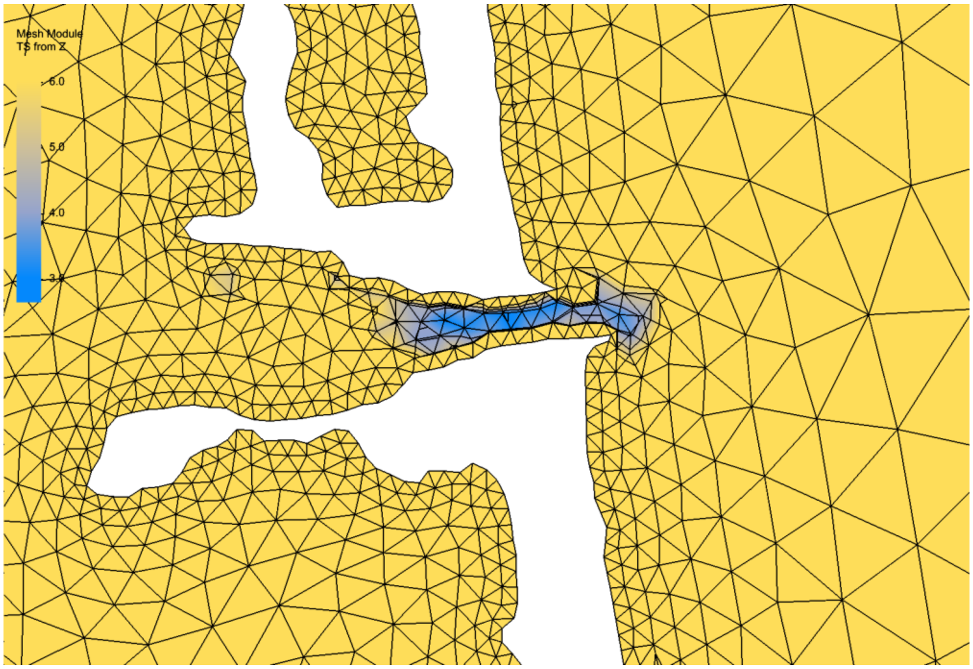
SMS 14.0 Tutorial

ADCIRC CFL Check



Objectives

This tutorial demonstrates how to use the Courant-Friedrichs-Lewy (CFL) condition to determine an appropriate time step for an unstructured grid. The CFL limitation applies to explicit numerical solvers and is a relationship between the element size and the computational time step.

Prerequisite Tutorials

- Overview
- Map Module
- Raster
- Background Data
- Simulation Domain
- Mesh Generation
- Mesh Quality

Required Components

- SMS Core

Time

- 10–15 minutes

1 Introduction

This exercise illustrates how to use the Courant-Friedrichs-Lewy condition to determine an appropriate time step for a numeric simulation using tools in the Surface-water Modeling System (SMS).

This exercise utilizes data from the region around Indian River Inlet located on Delmarva Peninsula along the Maryland coast between Delaware Bay and Chesapeake Bay.

2 Getting Started

Start by importing a project file containing an existing mesh:

1. Launch the SMS application.
2. From the *Menu* bar, select the *File* | **Open…** menu item to bring up the *Open* dialog.
3. From the *Files of type* drop-down, select “Project Files (*.sms)”.
4. Navigate to the *\ADCIRC CFL Check\ADCIRC CFL Check\data files* folder.
5. Select the “IR_tides.sms” file.
6. Click the **Open** button to import the project and exit the *Open* dialog.

The initial project should appear similar to Figure 1.

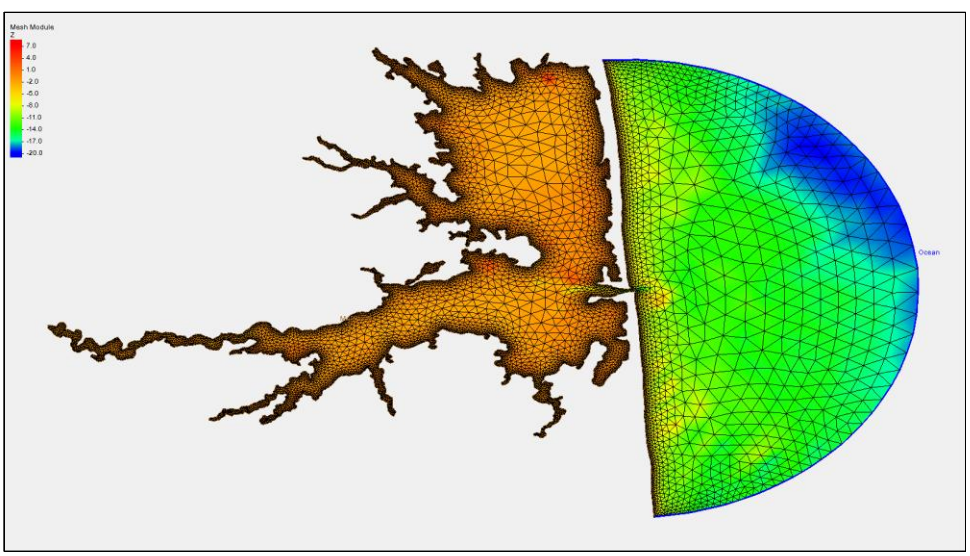


Figure 1: Initial Project

3 Computing an Allowable Time Step Dataset

The project loaded into SMS includes an unstructured grid. To apply this with an explicit numerical solver such as ADCIRC, a suitable time step must be selected. One way to conceptualize an explicit solver is to imagine it tracking individual water particles as they move through the domain. During a single computational time step, no water particle should traverse an entire mesh element, as doing so could result in inaccurate representation of local flow dynamics. Therefore, the time step must be small enough to ensure that particle movement remains within reasonable bounds for each cell, preserving the fidelity of the simulation.

The Courant number (*C*) can be computed using the following equation:

$$C = \frac{u\,\Delta t}{\Delta x} \leq C_{\max}$$

Where:

- C = Courant number (dimensionless)
- u = characteristic velocity (m/s)
- Δt = time step (s)
- Δx = characteristic element length (m)

The characteristic velocity used in the Courant number calculation can be obtained from previously computed results (e.g., ADCIRC output), if available. However, when initiating a new simulation, the characteristic velocity can be approximated using the speed of a gravity wave in water. This speed is given by the expression:

$$v = \sqrt{g} \times h$$

Where:

- v = velocity
- g = acceleration due to gravity (9.81 m/s²)
- h = local water depth

If the Courant number C is equal to 1.0, this indicates that the water particle has travelled exactly Δx in one time step—placing it right at the CFL (Courant–Friedrichs–Lewy) stability limit. However, Δx varies spatially across the mesh. Also, in two dimensional domains, water may traverse only part of a cell (e.g., through a corner), but the underlying stability principle still holds.

A common guideline in ADCIRC is to set the Courant number to 0.6 and calculate the corresponding time step for each node using local depth and element size. The smallest of these Δt values would then be the limiting value.

Note: You can observe that as the depth increases, the velocity also increases. Therefore, as the depth increases, the allowable timestep will decrease. In the provided mesh, the Δx increases with distance from the shoreline, which can permit larger time steps, assuming the depth does not increase proportionally.

3.1 Setting Time Step Dataset

To compute a time step dataset:

1. From the *Macro* tool bar, click on the **Toolbox** macro to open the *Toolbox (beta)* dialog.

2. Under the *Tools* tab, expand the “Datasets” folder, select the “Gravity Waves Time Step” tool and click the **Run Tool…** button to open the *Gravity Waves Time Step* dialog.
3. From the *Input dataset* drop-down, select the “Mesh Data/constant pave 50-1000m/Z” option.
4. For the *Use courant number* field, enter “0.6”.
5. For the *Gravity waves time step dataset* field, enter “TS for C = 0.6”.
6. Click the **OK** button to execute the tool and show the *Status* window.
7. For this example, should a warning appear in the log, it should not be significant and can be ignored.
8. Click the **OK** button to close the *Status* window and load the new dataset.
9. Click the **Close** button to exit the *Toolbox (beta)* dialog.

The mesh should appear similar to Figure 2.

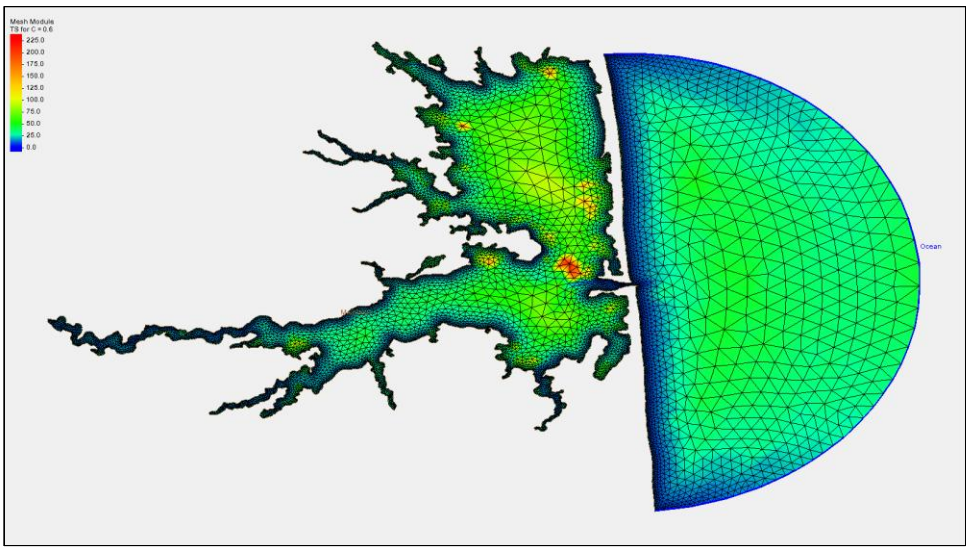


Figure 2: Computed Time-step Dataset

4 Reviewing Dataset Properties

A dataset of time-step values has been created. The objective of this exercise is to identify a numerically stable time step for the mesh. To determine this value:

1. In the *Project Explorer*, in the *Mesh Data* folder, right-click on “TS for C = 0.6” dataset and select the **Properties…** context menu item to open the *Properties* dialog.

Note the *Minimum* and *Mean* values. The minimum value represents the most restrictive time step needed for numerical stability anywhere in the mesh. The mean value can provide context but should not be used alone for setting the simulation time step.

2. Click the **OK** button to exit the *Properties* dialog.

5 Reviewing Time Step Sensitivity

To illustrate the sensitivity of the time step, repeat the procedure using a Courant number of 0.7 instead of 0.6. To do this:

1. From the *Macro* bar, select the **Toolbox** macro to open the *Toolbox (beta)* dialog.
2. Under the *History* tab, open the folder for the current date to display the tool executed above.
3. Select the tool and click the **Run Tool From History…** button to open the *Gravity Waves Time Step* dialog.
4. For the *Use courant number* field, enter “0.7”.
5. For the *Gravity waves time step dataset* field, enter “TS for C = 0.7”.
6. Click the **OK** button to execute the tool and open the *Status* window.
7. For this example, should a warning appear in the log, it should not be significant and can be ignored.
8. When the tool run is complete, click the **OK** button to close the *Status* window and load the new dataset.
9. Click the **Close** button to exit the *Toolbox (beta)* dialog.
10. In the *Project Explorer*, in the “Mesh Data” folder, right-click on “ TS for C = 0.7” dataset and select the **Properties…** context menu item to open the *Properties* dialog.

Review the *Minimum* and *Mean* values.

11. Click the **OK** button to exit the *Properties* dialog.

Note: The table below includes the approximate minimum and mean time step values corresponding to Courant numbers ranging from 0.6 to 1.0.

Courant #	Minimum time step	Mean time step
0.6	2.706	23.245
0.7	3.157	27.119
0.8	3.608	30.993
0.9	4.059	34.867
1.0	4.510	38.741

6 Using Display Theme

Use a display theme to refer to a particular location and contours.

1. From the *Menu* bar, select the *File* | **Open…** menu item to open the *Open* dialog.
2. From the *Files of type:* drop-down menu, select “All files (*.*)”.
3. Select the “IR_display_themes_13-4.xdt” file and click the **Open** button to close the *Open* dialog. A *Display Themes* folder appears in the Project Explorer window.

Note that this file contains a mesh contour options theme and a zoom theme focused on the inlet area. See the “Display Theme” tutorial for more information on using themes.

4. In the *Project Explorer*, in the *Mesh Data* folder, click on the “TS for C = 0.7” dataset.
5. In the *Project Explorer*, in the *Display Themes* folder, click on the “mesh-contours: TS Values” theme to update the contour options.
6. In the *Project Explorer*, in the *Display Themes* folder, click on the “Indian River Inlet Zoom” to zoom into the inlet.

The display should appear similar to Figure 3 Figure 1.

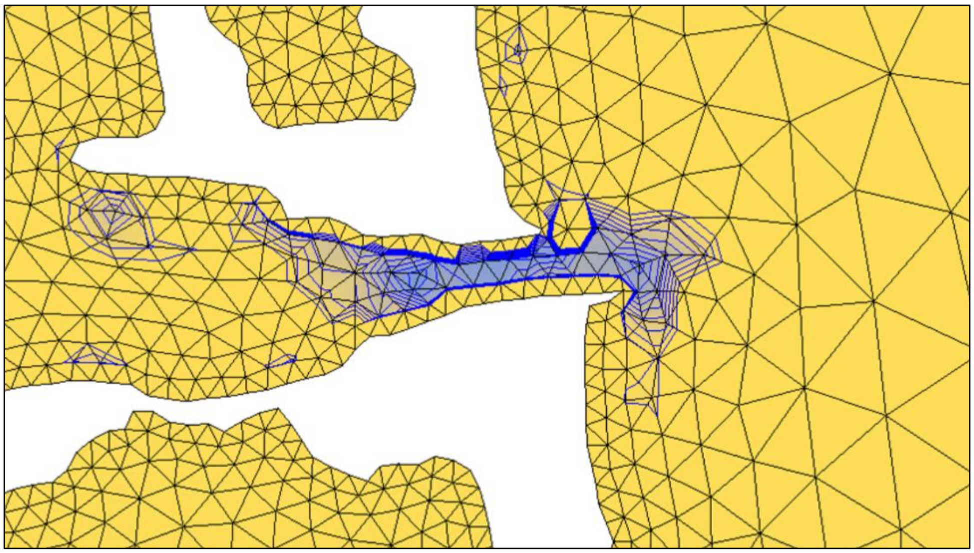


Figure 3: Contours of Timestep Values in the Inlet

This table and figure show that a small portion of the grid—the deep inlet region—limits the time step to a value between three and five seconds due to the combination of large depths and high mesh resolution (small elements) in that area.

7 Saving the Project

To save the project:

1. From the *Menu* bar, select the *File* | **Save As…** menu item to open the *Save As* dialog.
2. Navigate to the *\ADCIRC CFL Check\ADCIRC CFL Check\data files* folder.
3. For the *File name*, enter “IR_stability_check.sms”.
4. Click the **Save** button to exit the *Save As* dialog.

8 Conclusion

This concludes the “ADCIRC CFL Check” exercise. Several features have been demonstrated including:

- Computing a time step dataset from a specified Courant number
- Reviewing the properties of a dataset
- Invoking tools from the Toolbox history

