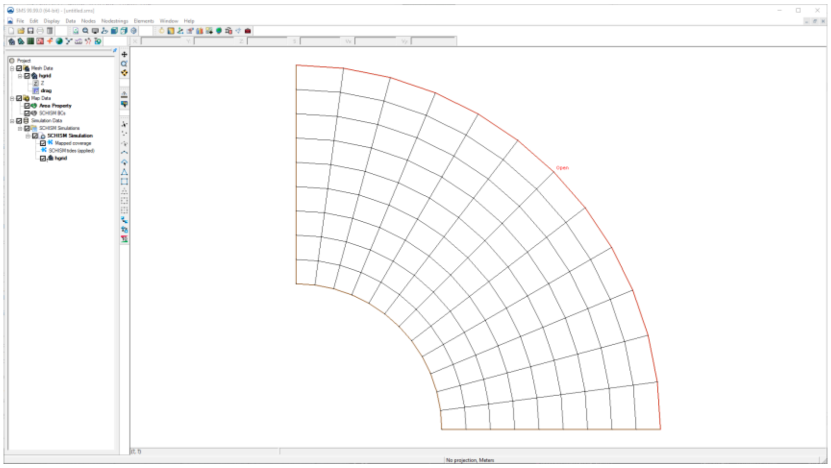
SMS 14.0 Tutorial

SCHISM – Quarter Annulus in SMS

Creating a SCHISM Simulation



Objectives

This tutorial demonstrates how to create a simple SCHISM simulation inside of SMS.

Prerequisite Tutorials

- Overview

Required Components

- SMS Core

Time

- 10–15 minutes

1 Introduction

This tutorial demonstrates setting up a SCHISM simulation inside of SMS.

SCHISM (Semi-implicit Cross-scale Hydro science Integrated System Model) is an open-source community-supported modeling system based on unstructured grids. Refer to https://ccrm.vims.edu/schismweb/ for more information.

SMS includes pre-processing tools for working with SCHISM. Most SCHISM runs require a supercomputer, so post-processing functionality is still in the works.

2 Getting Started

Start by opening the grid that will be used for this SCHISM simulation. (Refer to other tutorials to learn how to create a grid in SMS.) To open this file:

1. Launch the SMS application.
2. Select *File* | **Open…** to bring up the *Open* dialog.
3. Browse to the *data files* folder and select “quarter_annulus_grid.h5”.
4. Click **Open** to import the project and exit the *Open* dialog.

The project should appear similar to Figure 1.

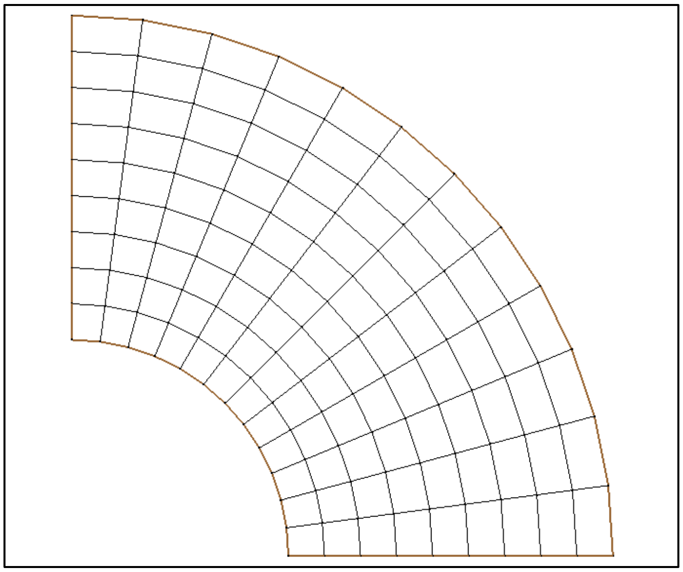


Figure 1 Initial grid

3 Creating a Simulation

The SCHISM approach in SMS is based on "simulation based modeling". This means that components, such as a mesh, can be reused in multiple simulations. This approach requires that a simulation entity exists in the SMS project. To create this:

1. Right-click on the background of the Project Explorer window and select *New Simulation* | **SCHISM (beta)** from the popup menu.
2. Right-click on the " Sim" simulation and select **Rename**.
3. Enter "qa_in_sms" for the new name.
4. Drag the " Quarter" mesh object to the " qa_in_sim" simulation to include this grid in the simulation. A line will appear under the simulation. When that line appears, let go of the mouse button to drop the mesh.

4 Boundary Conditions

The SCHISM simulation needs a boundary condition coverage to identify the open boundaries (locations along the edges of the domain where water can enter or leave the domain). Any boundary not specified as an open boundary will be assigned to be a closed boundary where no water can enter/leave the domain.

4.1 Creating the BC Coverage

To create the coverage:

1. Right-click on " Map Data" in the Project Explorer module and select **New Coverage** to open the *New Coverage* dialog.
2. Under *Coverage Type*, select *Models* | *SCHISM (Beta)* | **Boundary Conditions**
3. For the *Coverage Name,* enter "BC".

4. Click **OK** to close the *New Coverage* dialog

4.2 Defining the BC Arc

Next, define an arc that defines the open boundary by completing the following:

1. Use the **Create Feature Arc** tool to create an arc around the outer edge of the mesh (see Figure 2).

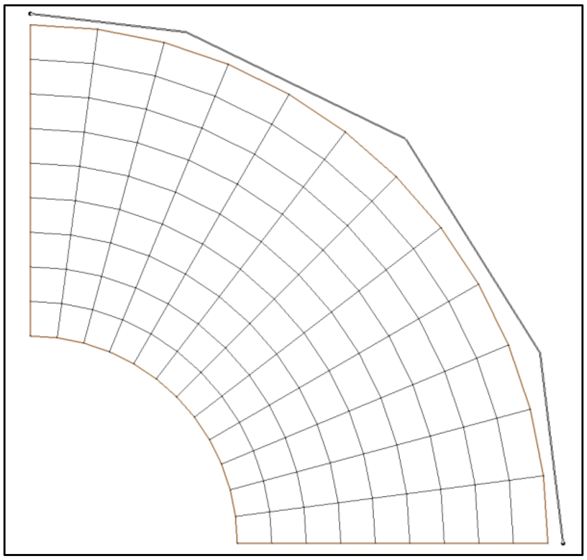


Figure 2 Position of the boundary condition arc.

This arc represents the open boundary. The position of this arc does not need to be precise. SMS will “snap” it to the outer boundary of the mesh. If the snapping does not match the desired open region, the arc can be edited later.

4.3 Assigning BC Attributes

To specify the type of open boundary condition:

1. Using the **Select Feature Arc** tool, double-click on the new arc to open the *Arc Properties* dialog.
2. For η: select option “3”.
3. For *Flow (u,v):* select option “3”.

This corresponds to tidal forcing.

4. For *Name (optional):* enter “Tide”.
5. Click **OK** to close the *Arc Properties* dialog.

4.4 Applying the BC Coverage

To apply the boundary condition coverage to the associated SCHISM simulation:

1. Right-click on the “BC” coverage and select *Apply to…* | **SCHISM Simulations → qa_in_sms.**

SMS maps the boundary conditions onto the mesh. (Note that if a mesh is not associated with the simulation, an error will be reported.) The mapped boundary conditions appear on the mesh. In the Graphics Window there will be two "Open" identifiers. The first identifies the open arc, while the second identifies how that arc snaps to the mesh.

5 Tidal Constituents

Since this simulation is being forced using tidal constituents, the desired constituents must be defined.

5.1 Loading the Local Tidal Values

In real world projects, tidal constituents are usually extracted from tidal databases. In theoretical cases such as this quarter annulus case, the user defines the tides. The tidal datasets must be defined on a mesh that covers the open boundary portion of the simulation mesh.

To read the tidal grid and datasets:

1. Select *File* | **Open…** to bring up the *Open* dialog.
2. Browse to the *data files* folder.
3. Hold the *Ctrl* key down a select both the "LocalTide.h5" and the "tide_dsets.h5" files.
4. Click **Open** to import the mesh and close the *Open* dialog.

Tidal components to forced water surface elevation consist of an amplitude and phase for each constituent. This can vary spatially, so it is defined as a dataset on a mesh. Forced velocities consist of two amplitudes and phases for the components of velocity.

5.2 Creating the Tidal Constituent Component

To create a tidal constituent component:

1. Right-click on the background of the Project Explorer window and select *New Simulation* | **Tidal Constituents** from the popup menu.
2. Right-click on the " Sim" simulation and select **Rename**.
3. Enter "UserTide" for the new name.

5.3 Defining the Tidal Constituent Properties

To assign the tidal properties:

1. Right-click on the " UserTide" simulation and select **Edit Constituents...** to open the *Tidal Constituents* dialog.
2. Change *Source* to "User defined".
3. Set the *Reference time* to "2/1/2000 12:00 AM"**.**
4. Click the **Add Row** button to add a constituent.
5. Change *Constituent* to "User specified" and enter these parameters:

   a. *Frequency*: "0.0001405257"
   b. *Nodal Factor*: "1.0"
   c. *Equilibrium Argument*: "0.0"
   d. *Force Velocity*: On
6. For the *Amplitude* in the constituent, click **(none select)** to open the *Select Tidal Dataset* dialog.
7. Select the "eta_amp" dataset and click **OK** to close the *Select Tidal Dataset* dialog.
8. Repeat steps 6–7 to assign the other datasets as follows:
   a. *Phase:* "phase_0"
   b. *Velocity Amplitude X*: "u_amp"
   c. *Velocity Amplitude Y*: "v_amp"
   d. *Velocity Phase X*: "phase_90"
   e. *Velocity Phase Y*: "phase_90"
9. Click **OK** to close the *Tidal Constituents* dialog and save the constituent definition

The set of constituent values were selected to match the values in the SCHISM SVN repository for this test case.

5.4 Applying the Tidal Constituents to the Simulation

To apply the tidal constituents the associated SCHISM simulation:

1. Right-click on the " UserTide" simulation and select *Apply to…* | **SCHISM Simulations → qa_in_sms**

Applying tidal constituents requires that the simulation know the open boundary locations in the simulation. Therefore, a tidal constituent cannot be successfully applied unless a boundary condition coverage has already been applied to the simulation.

6 Upwind Solver Options

If a SCHISM simulation is set with barotropic terms off or baroclinic terms on, the simulation requires an *Upwind Solver* specification provided. This is a flag for each cell in the mesh. This is specified using polygons in the SMS interface.

6.1 Creating the Upwind Solver Coverage

To create the coverage:

1. Right-click on " Map Data" in the Project Explorer module and select **New Coverage** to open the *New Coverage* dialog.
2. Under *Coverage Type*, select *Models* | *SCHISM* **| Upwind Solver Coverage**.
3. Accept the default name and click **OK** to close the dialog.

6.2 Defining the Upwind Solver Polygon

Next, define a polygon that covers the mesh by completing the following:

1. Select the " Upwind solver coverage" to make it active.
2. Use the **Create Feature Arc** tool to create an arc around the mesh. It should close on itself (see Figure 3).

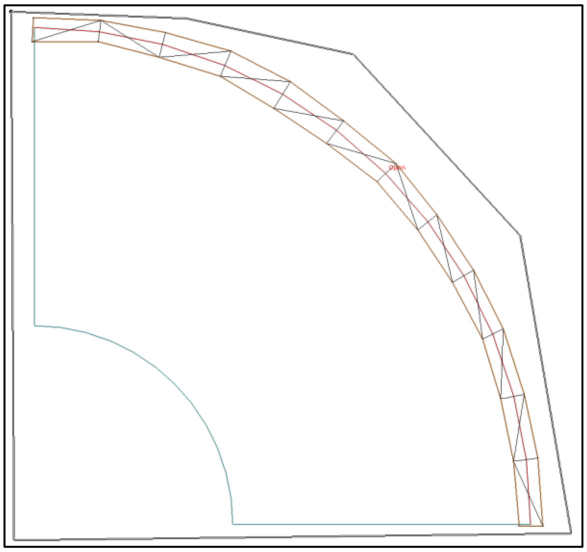


Figure 3 Upwind solver polygon.

3. Select the **Build Polygons** macro to build polygons.

For more complex simulations, there could be multiple polygons.

6.3 Assigning Upwind Attributes

To specify the type of open boundary condition:

1. Using the **Select Feature Polygon** tool, double-click in the new polygon to open the *Polygon Properties* dialog.
2. Turn on the *Upwind* option.
3. Click **OK** to close the *Polygon Properties* dialog.

6.4 Applying the Upwind Coverage

To apply the " Upwind solver coverage" to the associated SCHISM simulation:

1. Right-click on the " Upwind solver coverage" coverage and select *Apply to…* | **SCHISM Simulations → qa_in_sms.**

7 Creating the Drag Dataset

SCHISM requires a drag coefficient to be applied at each node in the mesh. If this value is spatially varied it can be created by interpolating from point data or assigned by polygon data. In this example, the drag is constant. However, it still needs to be a dataset. To create this dataset:

1. Click the **Toolbox** macro to open the *Toolbox* dialog.
2. In the *Search* field, enter “calculator”.
3. Select the “ Dataset Calculator” tool (in the “Dataset” folder) and click **Run Tool** to open the *Dataset Calculator Tool* dialog.
4. Select the following parameters for the tool:
    a. *Grid source selection:* “Mesh Data/Quarter”
    b. *Data location selection:* “Points”.
    c. *Mathematical expression:* “0.0”
    d. *Output dataset name:* “drag 0.0”
5. Click **OK** to run the tool.
6. After a moment, the top of the *Dataset Calculator* dialog will say “Successfully ran tool”. Click **OK** to close the dialog.

A new point based dataset named “ drag 0.0” has been created

8 Model Parameters

Now we are ready to assign the simulation parameters for this SCHISM simulation.

1. Right-click on the “ qa_in_sms” simulation and select **Model Control** to open the *SCHISM Model Cont*rol dialog
2. Enter the following parameters in the *CORE* tab:
    a. *Barotropic flag (ibc):* on
    b. *Baroclinic flag (ibtp):* off
    c. *Total simulation time in days (rnday)*: “5.0”
    d. *Time step in seconds (dt)*: “300.0”
    e. *Global output step spool (nspool)*: “3”
    f. *Global stack spool (ihfskip)*: “1440”
3. In the *OPT* tab
    a. *Start Month (start_month)*: “2”
    b. Under *Drag coefficient dataset:* Click on **Select…** to open the *Select dataset for Drag coefficient dataset* dialog.
        i. Select “drag 0.0”.
        ii. Press **OK** to close the *Select dataset for Drag coefficient dataset* dialog.
    c. *dfv0*: “0.0”
4. In the *SCHOUT* tab, click the **Edit Table…** button to open the *Global output options (iof_hydro)* dialog.

5. Turn on the *iof_hydro(16) Depth-averaged vel vector [m/s](depthAverageVelX,Y)* option.
6. Click **OK** to close the *Global output options* dialog.
7. In the *OTHER* tab
    a. *Cut-off depth for applying tidal potential (i.e., it is not calculated when depth < tip_dp:* "40"
    b. *Z levels:* set 1 value of 1,000,000 in the table.
        i. For the *Z levels*, click the **Edit Table…** to open the *Z levels* dialog.
        ii. Click the **Add Row** button and enter a value of "1000000".
        iii. Click **OK** to close the *Z levels* dialog.
    c. *hc:* "40.0"
    d. theta_b: "1.0"
    e. theta_f: "0.0001"
    f. *S levels:* set 2 values of 1.0 and 0.0 in the table.
        i. For the *S levels*, click the **Edit Table…** to open the *S levels* dialog.
        ii. Click the **Add Row** button and enter a value of "1.0".
        iii. Again, click the **Add Row** button and enter a value of "0.0".
        iv. Click **OK** to close the *S levels* dialog.
        v. Click **OK** to close the *Model Control* dialog.

9 Saving the Project

This project is now complete and ready to save.

9.1 Saving the Simulation

To save this new SMS project and the associated SCHISM simulation:

1. Select *File* | **Save New Project…** to bring up the *Save* dialog.
2. For the *File name*, enter "schism_qa".
3. Click the **Save** button to close the *Save* dialog.
4. Right-click on " qa_in_sms" and select **Save Simulation**

SMS has created a *schism_qa_models* folder next to the "schism_qa" project. In this folder it has created a *SCHISM/qa_in_sms* folder that contains the SCHISM simulation files listed below:

- bctides.in
- drag.gr3

- hgrid.gr3
- param.nml
- vgrid.in

These can be compared with the files of the same name in the folder from the SCHISM repository.

9.2 Running the Simulation

To run the simulation:

1. Right-click on " qa_in_sms" and select **Run Simulation**.

SMS will launch the specified SCHISM executable using the input files created in the previous section. This will result in a set of outputs for the simulation. In the future, SMS will be able to load the outputs for post processing.

10 Conclusion

This concludes the "SCHISM Quarter Annulus in SMS" tutorial.

Refer to other SCHISM tutorials or documentation to learn about model options and output.

