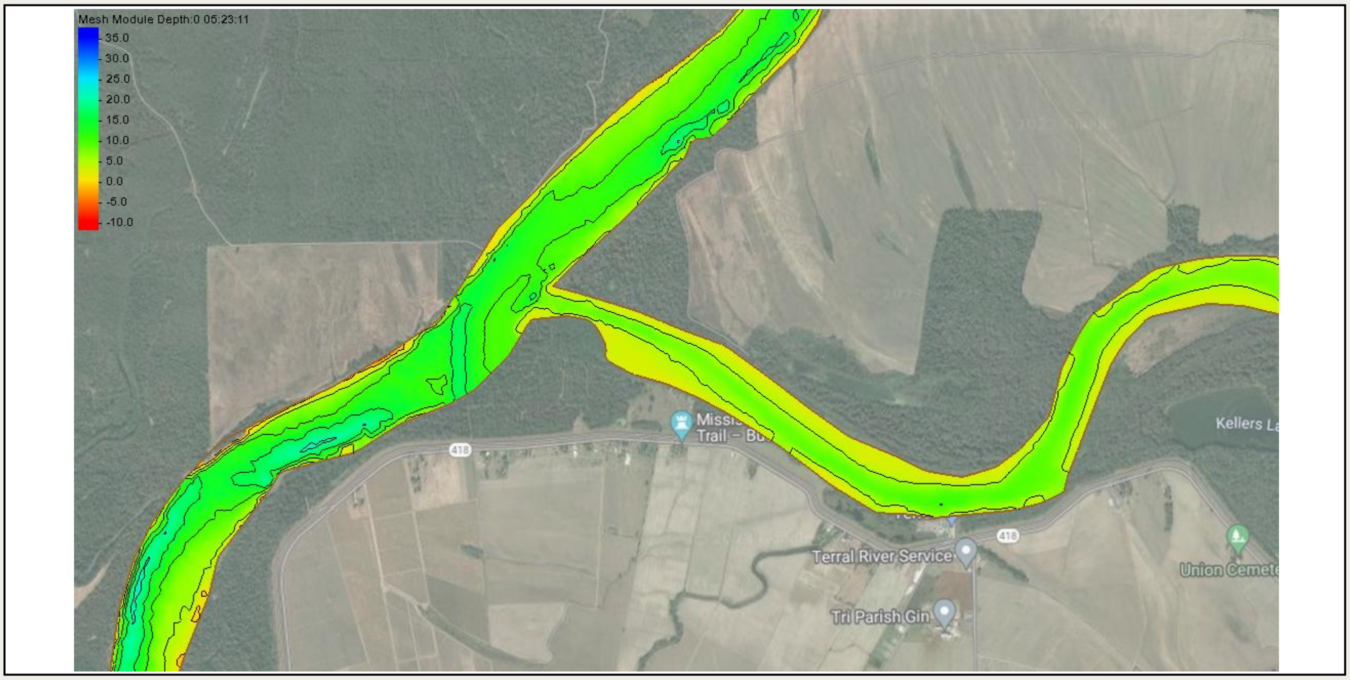
SMS 14.0 Tutorial

Adaptive Hydraulics (AdH)

Create a 2D AdH model using SMS



Objectives

This tutorial shows how to use the interface for AdH and run the model for a sample application. This example steps through the process of setting up and running a simulation using data from the area of the Red River.

Prerequisite Tutorials

- Overview
- Map Module
- Mesh Generation

Required Components

- SMS Core
- AdH Model & Interface

Time

- 15–30 minutes

AQUAVEO™

1 Introduction

Adaptive Hydraulics (AdH) is a multi-dimensional modeling system for saturated and unsaturated groundwater, overland flow, three-dimensional Navier-Stokes flow, and two-dimensional or three-dimensional shallow water problems. The 2-dimensional (2D) shallow water module of AdH was developed by the Coastal and Hydraulics Laboratory at the Engineer Research and Development Center in Vicksburg, MS. To learn more about the AdH model, visit their website:
https://www.erdc.usace.army.mil/Locations/CHL/AdH/

This tutorial will demonstrate how to set up a new AdH model in SMS. The tutorial will make use of example project located in a section of the Red River in Louisiana near the Mississippi River Northwest of Baton Rouge. The example project contains a 2D mesh with an elevation dataset, a material coverage, and a boundary condition coverage. The 2D mesh and the arcs for the coverages are provided to minimize time required to run this tutorial. See the “Map Module” and “Mesh Generation” tutorials for the process used to generate the starting project.

2 Getting Started

The mesh and coverage used for this tutorial were previously created. Open the background data for this project by doing the following:

1. If necessary, launch SMS.
2. If SMS is already running, select *File* | **New** to ensure that the program settings are restored to their default state.
3. Select *File* | **Open…** to bring up the *Open* dialog.
4. Browse to the *data files* folder for this tutorial and select “RedRiver.sms”.
5. Click **Open** to import the project and exit the *Open* dialog

The project should appear similar to Figure 1. Before continuing, it is best to save the project under a new name in order to preserve the starting project for future use.

6. Select *File* | **Save As…** to bring up a *Save As* dialog.
7. Enter the *File name* “RedRiverAdH.sms” and click **Save** to close the *Save As* dialog.

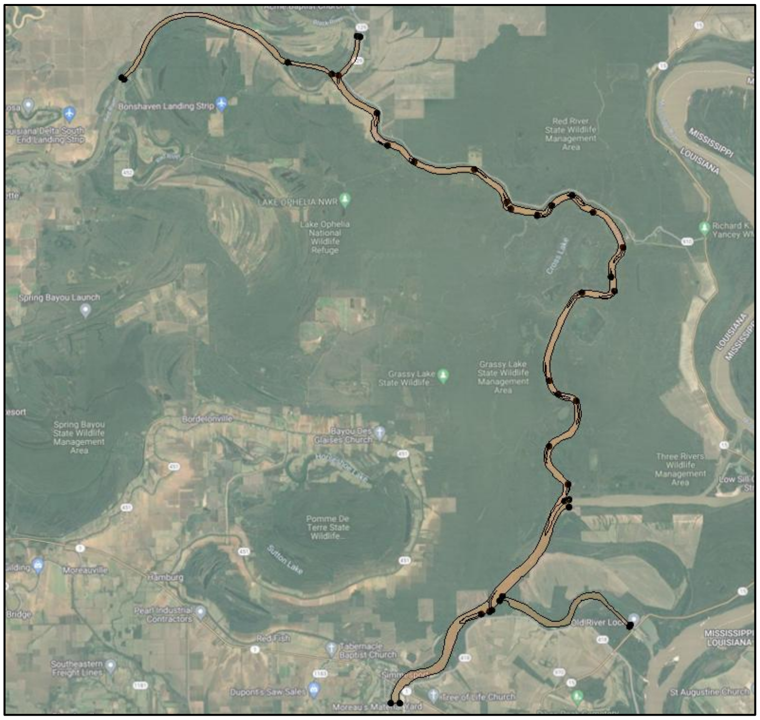


Figure 1   Initial project

3 Assign Material Polygons

Start with assigning the materials to the polygons on the materials coverage. This example will make use of two materials which will have similar properties. Material 1 will be applied to the shallow areas and material 2 will be applied to the deeper areas.

1. Select the " Red River Materials" coverage to make it active.
2. Right click on the " Red River Materials" coverage and set its type to *Models* | *AdH* | **Materials**.
3. Right-click on the " Red River Materials" coverage and select **Material List and Properties** to open the *Material List and Properties* dialog.
4. Click the **Add Row** button twice to create two new material types.
5. Select "Material 1" then select the *Friction* tab.
6. Change the *Friction* drop-down to "Manning's N (MNG)".
7. Change the *Manning's n* edit field to "0.025".
8. Repeat steps 5–7 for "Material 2".
9. Click **OK** to exit the *Material List and Properties* dialog.
10. Using the **Select Feature Polygon** tool, double-click inside the river polygon that extends the length of the model to open the *Assign Material* dialog.
11. Change the selection to "Material 1" and click **OK** to close the *Assign Material* dialog.
12. Without unselecting the polygon, right-click in the Graphics Window and select **Invert Selection**.

The info bar at the bottom of the screen should state that 25 polygons are selected.

13. Right-click and select **Assign Material** to open the *Assign Material* dialog.
14. Change the selection to “Material 2” and click **OK** to close the *Assign Material* dialog

There are 25 smaller polygons which should be assigned to “Material 2”.

4 Assign Boundary Conditions

AdH requires assigning boundary conditions to provide inlet discharge to the river and hydraulic properties of the exit discharge of the model. There are three locations where water flows into the model and one location where water is discharged from the model. The locations are shown in Figure 2. Red River, Lock and Dam #1, Black Bayou at Acme, Louisiana, and the Old River Control Structure are inflow boundaries while the Simmsport defines the downstream end of the simulation where flow leaves the domain.

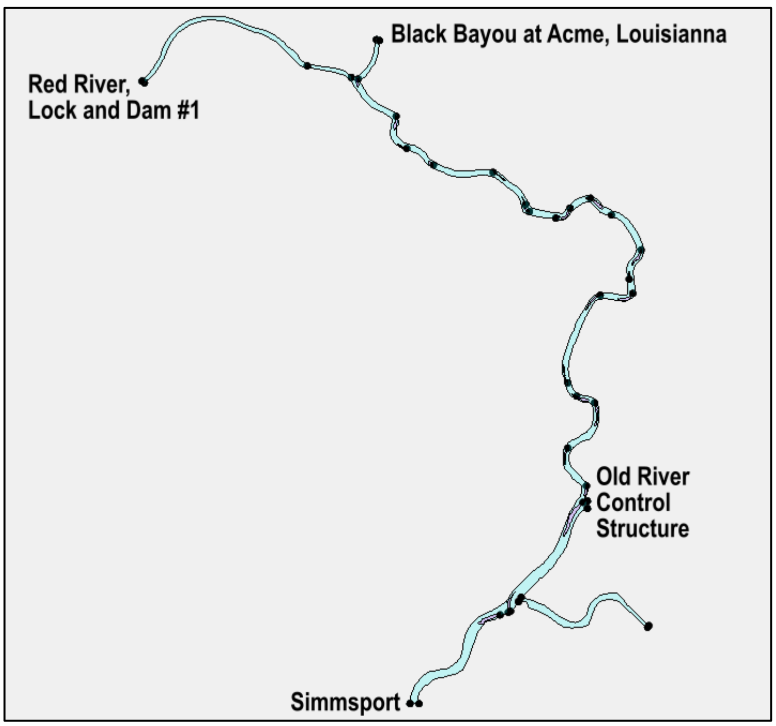


Figure 2 Boundary Condition arcs with locations labeled

1. Turn off the “Red River Materials” coverage.
2. Select the “Red River Boundary Conditions” coverage,
3. Right-click on the “Red River Boundary Conditions” coverage and set its type to *Models* | *AdH* | **Boundary Conditions**.
4. Using the **Select Feature Arc** tool, double-click on the arc at the upstream end of the river labeled “Red River, Lock and Dam 1” in Figure 2 to open the *Arc Attributes* dialog.
5. Set the *Arc type* to “Discharge (NB DIS)” and click the **Edit** button to open the *XY Series Editor* dialog.
6. Click the **Import** button to bring up the *Open* dialog.

7. Browse to the *data files* folder for this tutorial and select the “L&D_1.xys” file. Note that this file includes flow rates in cubic meters per second verses time in seconds.
8. Click **OK** to close the *XY Series Editor* dialog.
9. Click **OK** to close the *Arc Attributes* dialog.
10. Repeat steps 4–9 for the “Black Bayou at Acme, Louisiana” location and selecting the “Acme.xys” file in step 7.
11. Repeat steps 4–9 for the “Old River Control Structure” location and selecting the “ORCS.xys” file in step 7.
12. Using the **Select Feature Arc** tool, double-click on the arc labeled “Simmsport” in Figure 2 to open the *Arc Attributes* dialog.
13. Set the *Arc type* to “Water surface elevation (NB OTW)” and click the **Edit** button to open the *XY Series Editor* dialog.
14. Click the **Import** button to bring up the *Open* dialog.
15. Browse to the *data files* folder for this tutorial and select the “Simmsport.xys” file. Note that this file contains water levels in meters vs time in seconds.
16. Click **OK** to close the *XY Series Editor* dialog.
17. Click **OK** to close the *Arc Attributes* dialog.

5 Create and Setup the AdH Simulation

The next step in preparing a simulation is to specify model input parameters by first creating the AdH simulation:

1. Right-click on an empty space in the Project Explorer and select *New Simulation* | **AdH**.
2. Right-click on the newly created “ Sim” item and select **Rename**.
3. Enter “RedRiver” and press *Enter* to set the new name.
4. Drag “ Red River AdH Mesh”, “ Red River Materials”, and “ Red River Boundary Conditions” under “ RedRiver”.
5. Double-click on the “ RedRiver” simulation to open the *Model Control* dialog.

5.1 Set Time Options

Start with setting the options on the *Time* tab.

1. Change the *End Time:* to “10.0” and *End Time units:* to “3 – days”.
2. Change the *Time step option:* to “Time step series (Series DT)”.
3. Below *Max time step size time series*, click the **Edit Series** button to open the *XY Series Editor* dialog.
4. Increase the *Number of rows* to “3”.
5. Enter the following data into the *X* and *Y* columns:

0.0	400.0
1440.0	1800.0
8647000.0	3600.0

6. Click **OK** to close the *XY Series Editor* dialog.

5.2 Set Iteration Options

Now define the iterations by doing the following:

1. Select the *Iteration* tab.
2. Change *Non linear iterations*: to “6”.
3. Change the *Non linear residual tolerance*: to “0.1”.
4. Change the *Linear iterations:* to “200”.

5.3 Set Operation Options

Now define the simulations operations by doing the following:

1. Select the *Operation* tab.
2. Turn off the *Second order temporal coefficient active* option if it is not already off.
3. Turn off the *Petrov Galerkin coefficient active* option if it is not already off.

5.4 Set Output Options

Now set the output options by doing the following:

1. Select the *Output* tab.
2. Change the *Output control option:* to “Specify autobuild (OS)”.
3. Change the *End Time* to “240”.
4. Change the *Interval* to “1.0”.
5. Change the *Unit* to “hours”.
6. Turn on the *Screen output all* option.

5.5 Set Constants Options

Now define the simulation constants by doing the following:

1. Select the *Constants* tab.
2. Change the *Wet dry stabilization length:* to “0.1”.

5.6 Set Hot Start Option

Now define the hot start options by doing the following:

1. Select the *Hot Start* tab.
2. Set the first (0) *dataset* to be “”ioh”.
3. Click **OK** to close the *Model Control* dialog.

6 Saving and Running the AdH Simulation

The model is now set up with geometry, material & boundary condition definitions, and the simulation settings configured. The model can now be run. Run times will vary, though the model will take significant time to complete.

1. Right-click on the " RedRiver" simulation and select **Save Project, Simulation, and Run** to open the *Simulation Run Queue* dialog.
2. If a *Model Checker* dialog appears, there was a mistake in setting up the model. Fix the errors shown in the *Model Checker* dialog, save and run the model again.
3. If a message appears stating the coverages need to be renumbered, click **Yes**.

If using a standard installation of SMS, the model should launch immediately. If SMS cannot find the executables, a message will be displayed asking to locate the Pre-AdH executable and then the AdH executable.

This simulation may take several minutes to run, depending on the speed of the computer being used. If it takes more than an hour, then it is likely that there was a mistake in the model and it is not converging. Please review the tutorial and model to ensure that each step was followed.

4. After the model run is complete, click the **Load Solution** button.

7 Visualize Simulation Results

The model run created several solution scalar datasets.

- Depth SI
- Overland Velocity SI
- ERROR SI
- HYDRO_ERROR SI

SMS creates a folder in the Project Explorer containing all of the datasets (in this case, "RedRiver").

1. Select the " Depth SI" dataset to make it active.
2. Click on the **Display Options** macro to open the Display Options dialog.
3. Turn off the *Nodes, Nodal BC, Elements,* and *Elements BC* options.
4. Turn on the *Contours* option.
5. Click **OK** to close the *Display Options* dialog.
6. In the *Time steps* window, select different time steps.
7. Move around the mesh and review the results of the " Depth SI" dataset and the changes over time steps.
8. Click on the **Display Options** macro to open the Display Options dialog.
9. Turn on the *Vectors* option.
10. Click **OK** to close the *Display Options* dialog.
11. Select different time steps, move around the mesh, and review the results of the depth dataset and the changes over time steps.

Note that the first time step does not have any reported velocity.

12. Click on the **Display Options** macro to open the Display Options dialog.
13. Turn off the *Vectors* option.
14. Click **OK** to close the *Display Options* dialog.
15. Select the " ERROR SI" dataset and review the results.

Note that there is no reported error on the first time step.

16. Select the " HYDRO_ERROR SI" dataset and review the results.

Note that there is no reported error on the first time step.

All of the standard visualization methods described in the "Data Visualization" tutorial also works for the solutions generated by AdH. Experiment with other options to view the solution.

8 Conclusion

This concludes the AdH tutorial. Feel free to continue experimenting with the SMS interface or exit the program.