SURFACE WATER MODELING SYSTEM

TUFLOW – Advection Dispersion

1 Introduction

This tutorial describes the generation of a TUFLOW Advection Dispersion project using the *SMS* interface. The TUFLOW Advection Dispersion module is used for tracking constituent into a bay, and salinity intrusion.

More information about TUFLOW can be obtained from the TUFLOW website: www.tuflow.com.

2 Background Data

SMS modeling studies requires or uses several types of data. This data includes:

1. Geographic (location) and topographic (elevation) data. Note that all units in TUFLOW are metric.

2. Land use data (may be extracted from images or read as map file)

3. Boundary conditions.

We will start by loading the first item of data.

2.1 Bathymetry Data

Topographic data in SMS is managed in the scatter module as scattered data sets or triangulated irregular networks (TIN). SMS uses this data as the source for elevation data in the study area. To open the scattered data:

1. Select *File* | *Open* and open the file *madora_ad.sms* from the Data Files Folder for this tutorial.

The screen will refresh, showing a set of scattered data points as well as Boundary Conditions coverage as shown in Figure 1.

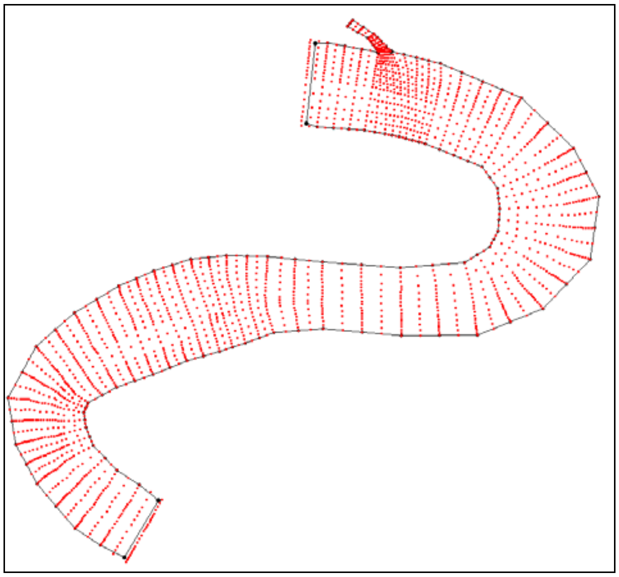

Figure 1 Scatter data

2.2 Modifying the Display

Now that our initial data is loaded, let's adjust the display

Settings

Make sure the following display settings are being used.

1. Choose *Display* | *Display Options*.
2. In the *Scatter* tab, make sure *Points* are turned off and the TIN *Boundary* and *Contours* are turned on.
3. In the *Contour Options* tab, set the *Contour Method* to *Color Fill* and set the *Transparency* to 50%. Click OK.

3 Creating the 2D Model Inputs

A TUFLOW model uses grids, feature coverages, and model control objects. In this section we will build the base grid and coverages. We will add model control information and additional objects later.

3.1 TUFLOW Grid

We need to create a TUFLOW Grid, and in order to do so, we need to create a new coverage.

1. Right click on *Map Data* in the *Project Explorer* and select *New Coverage* from the drop down menu.

2. Name the new coverage "TUFLOW Grid" and change its type to *TUFLOW | 2D Grid Extents*. Click *OK*.

3. Select the new coverage named "TUFLOW Grid" to make it active. Using the *Create 2-D Grid Frame* tool, create a grid frame as shown in Figure 2.

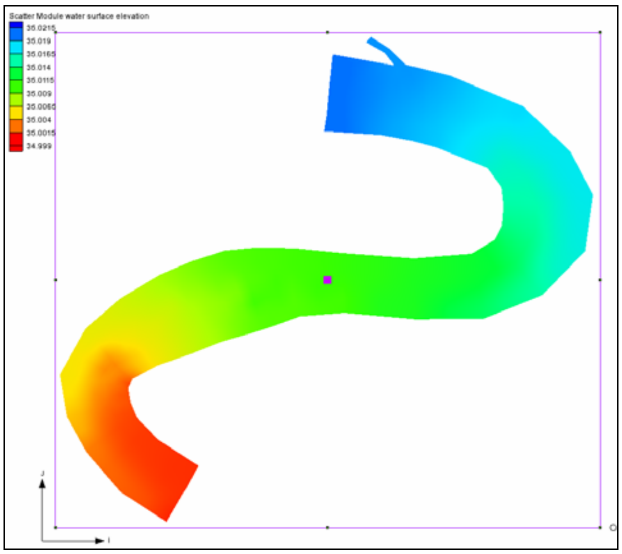

Figure 2 Grid Frame

4. If you wish to edit the location/size of the grid frame after creating it, first select the grid frame by choosing the *Select 2-D Grid Frame* tool and clicking in the box in the center of the grid frame. This exposes the editing handles. You can drag the handles on each side and corner of the grid frame to adjust the size of the grid frame. The circle near one of the grid frame corners can be used to rotate the grid frame.

5. Select *Feature Objects* | *Map->2D Grid*. This will bring up the *Map->2D Grid* dialog.

6. Set the C*ell Size* to 40 meters in the *I Cell Options*. This will also set the cell size in the *J Cell Options*.

7. In the *Elevation Options* section of the *Map -> 2D Grid* dialog change the *Source* to "Scatter Set", then click the "Select" button under it. Leave everything as default, except change the *Extrapolation Single Value* to 35 m. SMS assigns all cells not inside the TIN to this value. The value was chosen because it is

above all the elevations in the TIN, but not so large as to throw off the contour intervals.

8. Select *OK* twice. This will create a new item in the project explorer under *Cartesian Grid Data* named "TUFLOWGridGrid". Rename the Cartesian grid to "40m_AD".

3.2 Area Properties

An area property coverage defines the material zones of your grid. This can be done by digitizing directly from an image or the data can be imported from an ESRI shapefile. SMS also supports reading the data from MapInfo mif/mid files.

TUFLOW can read the area property data from either GIS data or data mapped to the grid. We will use data mapped to the grid.

1. Open the map file "materials.map".

2. Click on "materials" in the Map Data tree in the Project Explorer to make it active. Using the *Select Feature Polygon* tool, select each polygon and make sure that they are all assigned a material property. The assigned material property of a selected polygon can be viewed by selecting *Feature Objects | Attributes…* or by selecting *Attributes…* from the right-click menu. New materials can be added using *Edit | Materials Data*.

We need to associate the area property with the grid. This is specified in the grid options dialog. At the same time, we will specify that the grid will use cell-codes from BC coverages. To do this:

1. Right click on the Cartesian Grid labeled "40m_AD" in the project explorer and select *Options…* from the drop down menu.

2. Under *Materials* select the radio button *Specify using area property coverage(s)*.

3. Change the *Default material* to "bank".

4. Under *Cell codes* select the radio button *Specify using BC coverage(s)*.

5. Change the *Default code* to "Inactive cell – not in mesh".

6. Click *OK* to exit the Grid Options dialog.

3.3 2D BC Coverage

We need to specify the boundary conditions for our model. This model will include a flow rate boundary condition on the upstream and side channel portions of the model and a water surface elevation boundary condition on the downstream portion of the model.

A boundary condition definition consists of a boundary condition category and one or more boundary condition components. TUFLOW supports the ability to combine multiple definitions into a single curve.

In addition to having multiple components, a boundary condition can also define multiple events. For example, it can store curves for 10, 50, and 100 year events in the same boundary condition. The event that will be used when running TUFLOW is specified as part of a simulation.

To assign boundary conditions for the main upstream arc (top arc of main channel):

1. Right click on the BC coverage in the Project Explorer and change the *Type* to *TUFLOW |1D/2D BCs and Links.*

2. Select the upstream arc using the S*elect Feature Arc* tool. Right click and select *Attributes*.

3. Change the type to *Flow vs Time (QT)*.

4. Click *Add/Remove Events* if SS is not in the *Events* window. Otherwise, skip to step 6.

5. Click *Add* and enter "SS" as the name. Click *OK* twice.

6. Select the "SS" event.

7. Click on the box currently labeled "Curve undefined" to bring up the *XY Series Editor* dialog.

8. Open the file "bc1.xls" in a spreadsheet program, and copy the times to the first column and the inflow values to the second column.

9. Click *OK* twice.

To assign boundary conditions for side channel:

1. Using the *Select Feature Arc* tool, double click the small side channel upstream arc. This will bring up an Attributes dialog.

2. Change the type to *Flow vs. Time (QT)*.

3. Select the "SS" event and click on currently labeled "Curve undefined" button.

4. Open the file "bc2.xls" in a spreadsheet program, and copy the times to the first column and the flow values to the second column.

5. Click OK twice to return to the main screen in SMS.

To assign boundary conditions for the downstream portion of the main channel (lower arc of main channel):

1. Using the *Select Feature Arc* tool, double click the downstream arc. This will bring up an Attributes dialog.

2. Change the type to *Wse vs Time (HT)*.

3. Select the "SS" event and click on currently labeled "Curve undefined" button.

4. Open the file "bc3.xls" in a spreadsheet program, and copy the times to the first column and the head (Wse) values to the second column.

Click *OK* twice to return to the main screen in SMS.

1. Make sure that the *BC* coverage is active by clicking on it.

2. Select *Feature Objects | Build Polygons*.

3. Using the *Select Polygons* tool, double click in the new polygon to bring up the *Polygon Attributes* dialog.

4. Change the *Type* to *Cell Codes* and change the *Code* to *Active*. Click *OK*.

4 TUFLOW Simulation

As mentioned earlier a TUFLOW simulation is comprised of a grid, feature coverages, and model parameters. We have created a grid and several coverages to use in TUFLOW simulations. SMS allows for the creation of multiple simulations each which includes "links" to these items. A link is like a shortcut in windows. The data is not duplicated; it just knows where to go to get the data. The use of links allows these items to be shared between multiple simulations. A simulation also stores the model parameters used by TUFLOW.

To create the TUFLOW simulation:

1. Right click in the empty part of the project explorer and choose *New Simulation | TUFLOW*. This will create several new folders that we will discuss as we go. Under the tree item named Simulations, there will be a new tree item named "Sim."

2. Rename the simulation tree item, "40m_SS_AD".

4.1 Geometry Components

Rather than being included directly in a simulation, grids are added to a "Geometry Component" which is added to a simulation. The geometry component includes a grid and coverages which apply directly to the grid.

Coverages that should be included in the geometry component include: *1D/2D BCs and Links* coverages (if they include code polygons), *2D Z Lines (advanced)* coverages, *2D Z*

Lines/polygons (simple) coverages, *2D Miscellaneous (FLC, WRF, IWL, and AD)* coverages, and *Area Property* coverages.

To create and setup the geometry component:

1. Right click on the folder named "Components" and choose *New 2D Geometry Component*.

2. Rename the new tree item from *2D Geom Component* to *40m*.

3. Drag under this tree item the grid named "40m_AD", the coverage named "materials", and the coverage named "BC".

4.2 Material Sets

Now that we have a simulation, we need to define our material properties. There is already a *Material Sets* folder, but we need to create material definition sets or a set of values for the materials.

1. Right click on the *Material Sets* folder and select *New Material Set*. A material set will appear below the *Material Sets* folder.

2. Right click on the material set in the project explorer and click *Properties* from the menu. The materials are displayed in the list box in on the left.

3. Change the values for Mannings n for the materials according to Table 1 Manning's n values. Click *OK* when done.

Table 1 Manning's n values

Bank	0.035
Channel	0.02
Overbank	0.05
Sandbar	0.055

4.3 Simulation Setup and model parameters

The simulation includes a link to the geometry component as well as to each coverage used that is not part of the geometry component. In our case all of the coverages in our simulation are part of the geometry component.

1. To create the link to the geometry component, drag the geometry component onto the simulation in the project explorer.

The TUFLOW model parameters include timing controls, output controls, and various model parameters. To setup the model control parameters:

2. Right click on the "40m_SS_AD" simulation and select *2D Model Control*.
3. Select the *Time* tab if it is not already selected. Set the *Start Time* to "0" hours and the *End Time* to "25" hours. Change the *time step* to "5.0" seconds. Change the Number of iterations to 2.
4. In the *Output Control* section, set the *Format Type* to *SMS 2dm*; the *Start Time* to "0" hours and the *Interval* to "600" seconds.
5. In the *Output Datasets* section, select the following datasets: *Depth*, *Water Level*, *Velocity Vectors*, and *Flow Vectors* (unit flowrate).
6. In the *Screen/Log Output* section, change the *Display interval* to *6*. While TUFLOW is running, it will write status information every 6 time steps.
7. Switch to the *Water Level* tab and change the *Initial Water Level* to "35.0 m".
8. Switch to the *BC* tab and switch the *BC Event Name* to *SS*.

Advection Dispersion

In the model control dialog, we also set up the Advection Dispersion data. To do so:

1. Switch to the *Constituents* tab and create a new constituent named "TSS".
2. Leave everything as defaulted except for the *Longitudinal Dispersion Coefficients* which should be changed to 12.6.
3. Click *OK* to close the *Model Control* dialog.
4. Using the *Select Feature Arc* tool, double click the upstream arc of the small side channel. This will bring up the Attributes dialog again.
5. In Options, click on the *Advection Dispersion BC…* button. This will bring up the *Advection Dispersion* dialog.
6. Select the constituent "TSS" and click on the currently labeled "Curve Undefined" button.
7. Open the file "TSS.xls" in a spreadsheet program, and copy the times to the first column and the concentration values to the second column. Click *OK*, *Done*, and *OK* to exit out of the dialogs.
8. Repeat steps 4 to 6 for the upstream and downstream arcs of the main channel. In the *XY Series Editor*, enter the following information for both cases:

Time (hours)	Concentrations
0	0.0
24	0.0

As can be seen, there are no constituents entering the network through the main channel and there are no constituents present at downstream boundary conditions initially. However the Advection Dispersion module requires that data for constituents is present at all the boundary conditions.

5 Saving a Project File

To save all this data for use in a later session:

1. Select *File | Save New Project*.
2. Save the file as “Madora_ad.sms”.
3. Click the *Save* button to save the files.

6 Running TUFLOW

TUFLOW can be launched from inside of SMS. Before launching TUFLOW the data in SMS must be exported into TUFLOW files. To export the files and run TUFLOW:

1. Right click on the simulation and select *Export TUFLOW files*. This will create a directory named TUFLOW where the files will be written. The directory structure models that described in the *TUFLOW users manual*.
2. Right click on the simulation and select *Launch TUFLOW*. This will bring up a console window and launch TUFLOW.

7 Using Log and Check Files

TUFLOW generates several files that can be useful for locating problems in a model. In the TUFLOW directory under \runs\log, there should be a file named “40m_SS_AD.tlf”. This is a log file generated by TUFLOW. It contains useful information regarding the data used in the simulation as well as warning or error messages.

This file can be opened with a text editor by using the *File | View Data File…* command in SMS.

1. Open the file “40m_SS_AD.tlf” by selecting *File | View Data File…*.

2. If the *View Data File* dialog appears, select “Notepad”(or some other text editor) and click *OK*.

3. Go to the bottom of the file.

The bottom of this file will report if the run finished, whether the simulation was stable, and report the number of warning and error messages. Some warnings and errors are found in the tlf file (by searching on ERROR or WARNING) and some are found in the messages.mif file (discussed below).

In addition to the text log file, TUFLOW generates a message file in .mif/.mid format. SMS can import mif/mid files into the GIS module for inspection. In the \runs\log directory, there should be a mif/mid pair of files named “40m_SS_AD_messages.mif”.

1. Open the file “40m_SS_AD_messages.mif” in SMS by selecting *File* | *Open…*.

2. In the *Mif/Mid import* dialog, make sure *GIS layer* is selcted and click *OK*.

This file contains messages which are tied to the locations where they occur. If your simulation had any ERRORS or WARNINGS, they will show up in this file. Otherwise the file will be empty.

It is sometimes difficult to read the messages because they are stacked on top of each other. You can use the info tool to see what the messages are. To use the info tool, click on the object (point at the start of the text string). This will bring up a dialog showing the attributes (in this case text) of the object or objects at the location.

The check directory in the TUFLOW directory contains several mif/mid files that can be used to confirm that the data in TUFLOW is correct. The info tool can be used with points, lines, and polygons to check TUFLOW input values.

8 Viewing the Solution

TUFLOW has several kinds of output. All the output data is found in a folder named “results” under the TUFLOW folder. Each file begins with the name of the simulation which generated the files.

The results folder contains a .2dm, .mat, .sup, and several .dat files including the “40m_SS_AD_TSS.dat” file which contains the simulation results for the constituents. These are SMS files which contain a 2D mesh and accompanying solutions, which represent the 2D portions of the model.

To view the solution files from within SMS:

1. Select *File* | *Open* from the menu bar. Open the *Results* folder from the TUFLOW directory.

2. Locate the “40m_SS_AD.xmdf.sup” file and open it. The TUFLOW output is read into SMS in the form of a two-dimensional mesh. If a dialog pops up and asks if you want to replace existing material definitions, click *No*. If a dialog pops up and asks for time units, select hours.

3. From the project explorer, turn off all Map Data, Scatter Data, and Cartesian Grid Data. Turn on and highlight the Mesh Data.

4. Open the *Display Options* dialog. From the 2D Mesh tab, turn on contours and vectors.

5. Switch to the Contours tab and select Color Fill as the contour method.

6. Click *OK* to close the *Display Options* dialog.

7. The mesh will be contoured according to the selected dataset and time step. At this point any of the techniques demonstrated in the post-processing tutorial can be used to visualize the TUFLOW results including filmloops and observation plots.

9 Comparing TUFLOW AD and RMA4 Solutions

RMA4 is part of the TABS-MD suite of programs and is used for tracking constituent flow in 2D model. An RMA4 simulation was run using the same information as in this TUFLOW AD tutorial and the results in both situations were compared.

Figure 3 shows the results obtained in the TUFLOW AD simulation at time “06:20:00”.

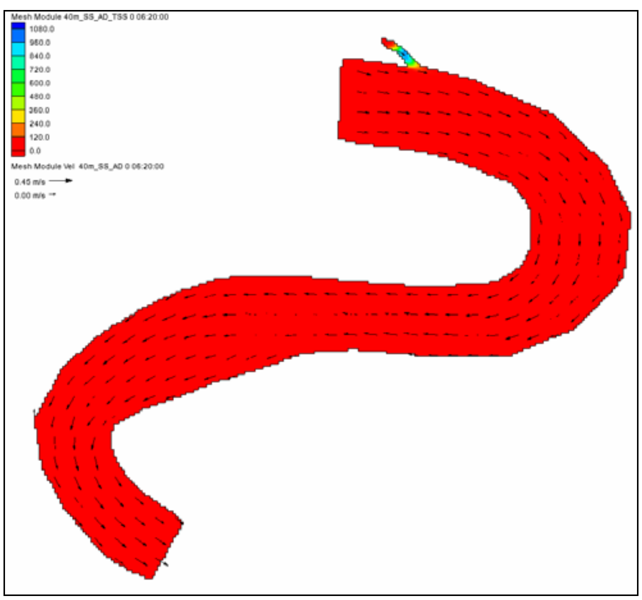

Figure 3 Results obtained in TUFLOW AD.

These results were compared to the results obtained in an RMA4 simulation. Figure 4 shows the result obtained in RMA4 at time “04:00:00”. This is about 2 hours ahead of the one obtained in TUFLOW AD. At time “06:20:00”, all the constituents were already washed out of the side channel in the RMA4 simulation.

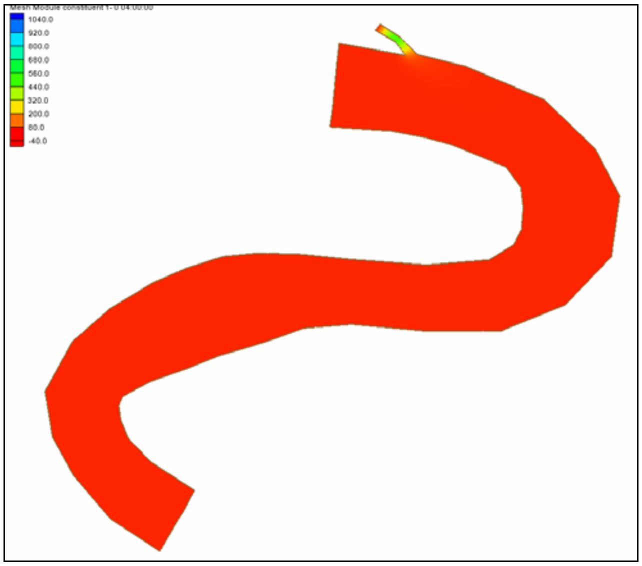

Figure 4 Results obtained in RMA4.

10 Conclusion

The results obtained when running TUFLOW AD is not completely the same as that obtained when running RMA4 even though the same original data were used. This is because the two models are different even though initial data is the same. The two models compute differently and some differences should be expected. These differences can be seen in the above pictures.

This concludes the *TUFLOW AD* tutorial. You may continue to experiment with the SMS interface or you may quit the program.

