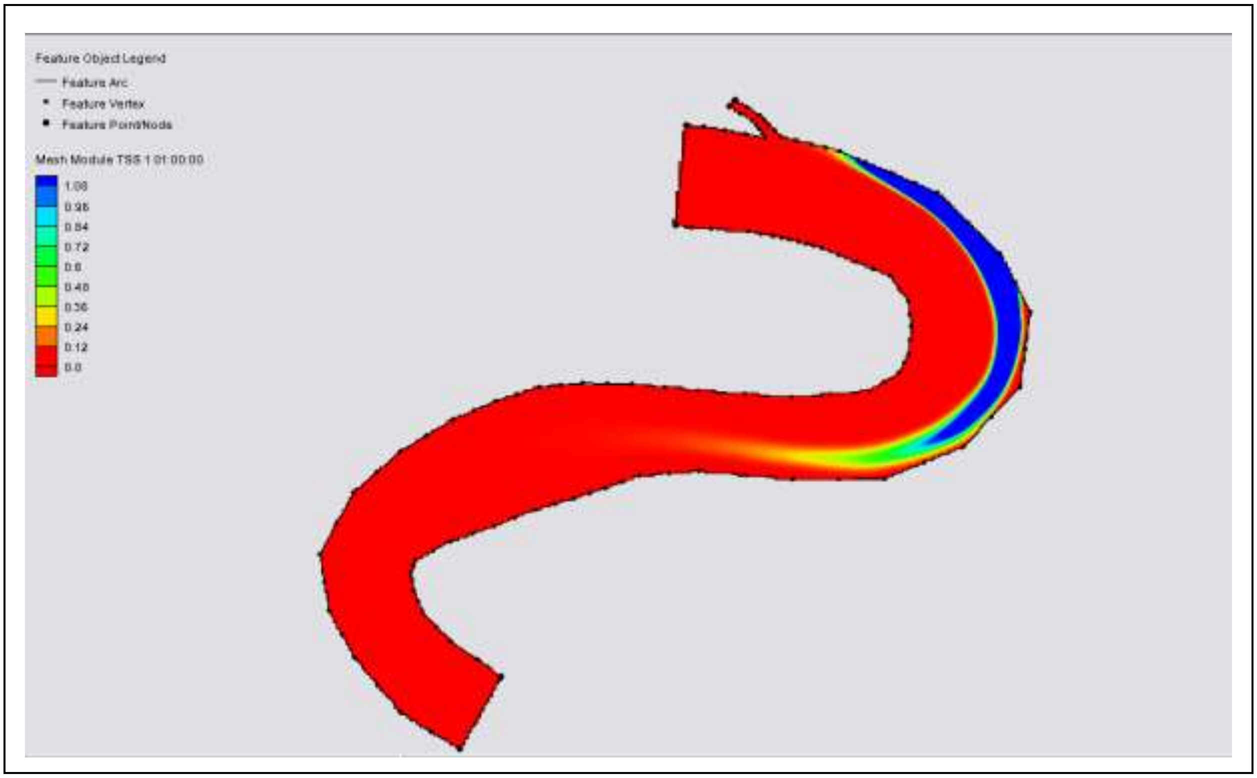
v. 11.2

SMS 11.2 Tutorial

TUFLOW - Advection Dispersion



Objectives

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 for salinity intrusion.

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

Prerequisites

- TUFLOW 2D
- Overview Tutorial

Requirements

- Map module
- Cartesian Grid Module
- Scatter Module
- TUFLOW-AD

Time

- 45-60 minutes

1 Background Data 2
1.1 Bathymetry Data 2
1.2 Modifying the Display 3
2 Creating the 2D Model Inputs 4
2.1 TUFLOW Grid 4
2.2 Area Properties 5
2.3 2D BC Coverage 5
3 TUFLOW Simulation 7
3.1 Geometry Components 7
3.2 Material Sets 8
3.3 Simulation Setup and model parameters 8
4 Saving a Project File 10
5 Running TUFLOW 10
6 Using Log and Check Files 11
7 Viewing the Solution 12
8 Comparing TUFLOW AD and RMA4 Solutions 12
9 Conclusion 14

1 Background Data

SMS modeling studies require or use several types of data, including:

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 a map file).
3. Boundary conditions.

Start by loading the first item of data.

1.1 Bathymetry Data

Topographic data in SMS is managed in the scatter module as scattered datasets 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 browse to the *data files* folder.
2. Open the file "madora_ad.sms".

The screen will refresh, showing a set of scattered data points as well as Boundary Conditions coverage as shown in **Error! Reference source not found.**.

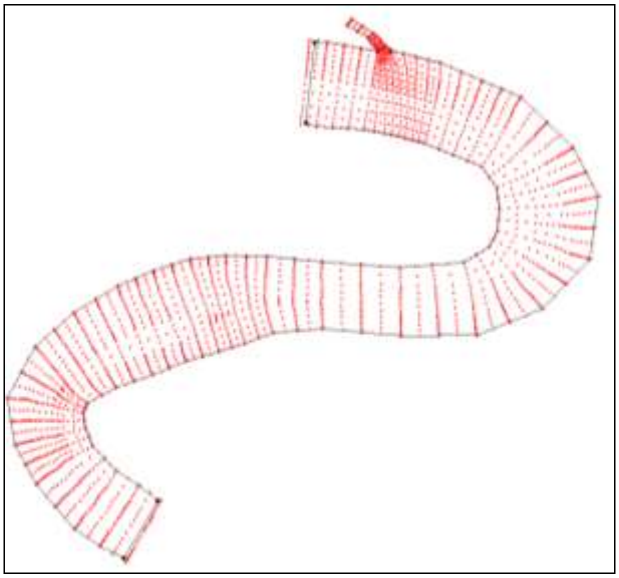


Figure 1 Scatter data

3. Select *Display* | **Projection…** and toggle on *Global Projection.*
4. Click on *Set Projection* and set:
    - Projection to “UTM”.
    - *Zone* to “15 (96 W - 90W - Northern Hemisphere)”.
    - *Datum* to “NAD83”.
    - Planar units to “METERS”.
5. Click **OK** twice when done.

1.2 Modifying the Display

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

Make sure the following display settings are being used.

1. Choose *Display* | **Display Options** to bring up the *Display Options* dialog.
2. Select “Scatter” from the list on the left.
3. On the *Scatter* tab, toggle off *Points*.
4. Toggle on Triangles, Boundary, and Contours.
5. On the *Contour* tab, set the *Contour Method* to “Color Fill” and set the *Transparency* to “50%”.
6. Click **OK**.

2 Creating the 2D Model Inputs

A TUFLOW model uses grids, feature coverages, and model control objects. In this section, the base grid and coverages will be built. Model control information and additional objects will be added later.

2.1 TUFLOW Grid

Create a TUFLOW Grid by first creating a new coverage using the following steps:

1. Right-click on “Map Data” in the Project Explorer and select **New Coverage** from the drop-down menu.
2. Set *Coverage Type* to *Models* | *TUFLOW* | **2D Grid Extents**.
3. Set *Coverage Name* to “TUFLOW Grid”.
4. Click **OK**.
5. Select the new coverage named “TUFLOW Grid” to make it active.
6. Using the **Create 2-D Grid Frame** tool, create a grid frame enclosing the entire S-curve, as shown in Figure 2.

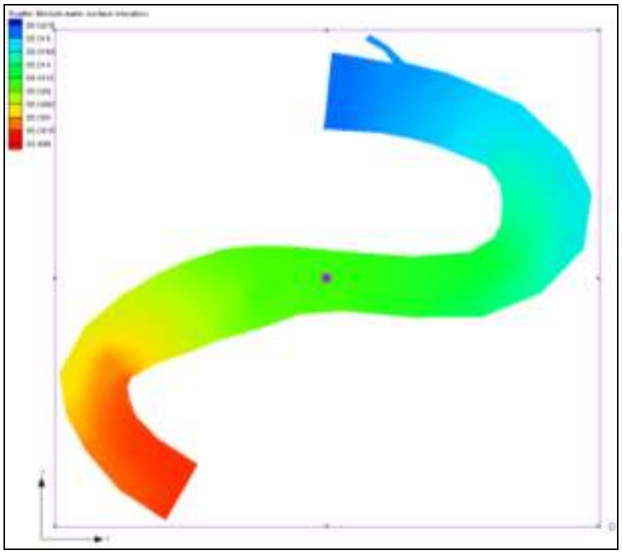


Figure 2 Grid frame

7. Select the grid frame by choosing the **Select 2-D Grid Frame** tool and click in the box in the center of the grid frame to expose the editing handles.
8. Drag the handles on each side and corner of the grid frame to adjust the size of the grid frame as necessary. The circle near one of the grid frame corners can be used to rotate the grid frame.
9. Select *Feature Objects* | **Map** → **2D Grid** to bring up the *Map* → *2D Grid* dialog.
10. Set the *Grid name* to “40m_AD”.

11. Set the C*ell Size* to 40 meters in the *I Cell Options* section. This will also set the cell size in the *J Cell Options*.
12. In the *Elevation Options* section, change *Source* to “Scatter Set”, then click the **Select** button under it.
13. Set *Extrapolation* to “Single Value”, then *Single Value* to “35”. SMS assigns this value to all cells not inside the TIN. 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. Leave all other settings at the default.
14. Select **OK** twice. This will create a new item named “40m_AD” in the Project Explorer under “Cartesian Grid Data”.

2.2 Area Properties

An area property coverage defines the material zones of the grid. This can be defined 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.

In this tutorial, data mapped to the grid will be used by doing the following:

1. Select *File* | **Open** and open the file “materials.map” in the *data files* directory.
2. Click on “materials” in the Map Data tree in the Project Explorer to make it active.
3. Using the **Select Feature Polygon**  tool, select each polygon and make sure that they are all assigned the appropriate 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**.

2.3 2D BC Coverage

The boundary conditions for the model must be specified. 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 select *Type* | *Models* | *TUFLOW* | **1D/2D BCs and Links**.

2. Select the upstream arc using the **Select Feature Arc** tool. Right-click and select *Attributes* to bring up the *Boundary Conditions* dialog.
3. Change *Type* to "Flow vs Time (QT)".
4. Click on Edit Events.
5. Click the **Add** button if "SS" is not in the *Events* window. This will add a "new_event" to the list. Double-click to rename it "SS".
6. Click **OK**.
7. Select the "SS" event.
8. Click on the large **Curve undefined** button to bring up the *XY Series Editor* dialog.
9. Open the file "bc1.xls" in a spreadsheet program.
10. Copy the time values from the *Time* column in the "bc1.xls" file to the *Time (hrs)* column in the *XY Series Editor* dialog.
11. Copy the flow values from the *Flow* column in the "bc1.xls" file to the *Flow (cms)* column in the *XY Series Editor* dialog.
12. Click **OK** twice.

To assign boundary conditions for side channel:

1. Using the **Select Feature Arc** tool, double-click the small upstream arc on the side channel to bring up the *Boundary Conditions* dialog.
2. Change the type to *Flow vs. Time (QT).*
3. Select the "SS" event and click on the large **Curve undefined** button to bring up the *XY Series Editor* dialog.
4. Open the file "bc2.xls" in a spreadsheet program.
5. Copy the time values from the *Time* column in the "bc2.xls" file to the *Time (hrs)* column in the *XY Series Editor* dialog.
6. Copy the flow values from the *Flow* column in the "bc2.xls" file to the *Flow (cms)* column in the *XY Series Editor* dialog.
7. Click **OK** twice.

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 to bring up the *Boundary Conditions* dialog.
2. Change the type to *Wse vs Time (HT).*
3. Select the "SS" event and click on the large **Curve undefined** button to bring up the *XY Series Editor* dialog.
4. Open the file "bc3.xls" in a spreadsheet program.
5. Copy the time values from the *Time* column in the "bc3.xls" file to the *Time (hrs)* column in the *XY Series Editor* dialog.

6. Copy the flow values from the *Wse* column in the “bc3.xls” file to the *Wse (m)* column in the *XY Series Editor* dialog.
7. Click **OK** twice.
8. Make sure that the “BC” coverage is active by clicking on it.
9. Select *Feature Objects* | **Build Polygons**.
10. Using the **Select Feature Polygon** tool, double-click in the new polygon to bring up the *Boundary Conditions* dialog.
11. Toggle on *Set Cell Codes*.
12. Select *Active* from the drop-down list.
13. Click **OK**.

3 TUFLOW Simulation

As mentioned earlier, a TUFLOW simulation is comprised of a grid, feature coverages, and model parameters. A grid and several coverages have been created to use in this tutorial. SMS allows for the creation of multiple simulations each which includes “links” to these items. The data is not duplicated. Instead, these items are 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 creates several new folders that will be discussed as the tutorial progresses. Under the tree item “Simulations” there is a new tree item named “Sim”.
2. Rename the new simulation to “40m_SS_AD”.

3.1 Geometry Components

Rather than being included directly in a simulation, grids are added to a “Geometry Component” which is then 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 (if they include code polygons).
- 2D Z Lines (advanced).
- 2D Z Lines/polygons (simple).
- 2D Miscellaneous (FLC, WRF, IWL, and AD).
- Area Property.

To create the geometry component:

1. Right-click on the “Components” folder in the Project Explorer and select **New 2D Geometry Component**.

2. Rename the new tree item from "2D Geom Component" to "40m".
3. Drag under this new tree item the grid named "40m_AD", the coverages named "materials" and "BC".

The area property needs to be associated with the grid. This is specified in the grid options dialog. At the same time, specify that the grid will use cell-codes from BC coverages.

To do this:

1. Right-click on the newly created "40m" geometry component and select **Grid Options** to bring up the *Grid Options* dialog.
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**.

3.2 Material Sets

The material properties need to be defined now that the simulation has been created. There is already a *Material Sets* folder, but the material definition sets, or a set of values for the materials, needs to be created.

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 new "Material Set (2)" and select **Rename** from the menu, changing the name to "Madora".
3. Right-click on the new "Material Set (2)" and select **Properties** from the menu. The different materials are displayed in the list box in on the left.
4. Under the "Single Value" drop-down list , change the values for Mannings *n* for the materials according to the table below.
5. Click **OK** when done.

Material	**Manning's *n***
Bank	0.035
Channel	0.02
Overbank	0.05
Sandbar	0.055

3.3 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 this case, all of the coverages in the simulation are part of the geometry component. The TUFLOW model parameters include timing controls, output controls, and various model parameters.

To set up the simulation and model control parameters:

1. Drag the "40m" geometry component onto the "40m_SS_AD" simulation in the Project Explorer. This creates a link to the geometry component.
2. Right-click on the "40m_SS_AD" simulation and select **2D Model Control** to bring up the *TUFLOW 2D Model Control* dialog.
3. Select *Time* from the list on the left and set the following:
   - *Start Time (hrs)* to "0".
   - *End Time (hrs)* to "25".
   - *Time Step (s)* to "5.0".
   - Number of iterations to "2".
4. Select *Output Control* from the list and set the following in the *Map Output* section:
   - *Format Type* to "SMS 2dm".
   - *Start Time* to "0" hours.
   - *Interval* to "600" seconds.
5. In the *Output Datasets* section, select the following datasets:
   - Depth.
   - Water Level.
   - Flow Vectors.
   - Velocity Vectors.
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. Select *Water Level* from the list on the left, and change the *Initial Water Level* to "35.0 m".
8. Select *BC* from the list and set *BC Event Name* to "SS".

The Advection Dispersion data needs to be set up as well:

9. Select *Constituents* from the list and enter "TSS" in the *Name* field.
10. Leave everything as the default except for *Longitudinal* under *Dispersion Coefficients*, which should be changed to "12.6".
11. Click **OK** to close the dialog.
12. Select the "BC" coverage to make it active.
13. Using the **Select Feature Arc** tool, double-click the upstream arc of the small side channel. This brings up the *Boundary Conditions* dialog again.
14. Click on the **Advection Dispersion BC…** button at the bottom of the dialog to bring up the *Advection Dispersion* dialog.

15. Select the constituent "TSS" and click on the large **Curve Undefined** button.
16. Open the file "TSS.xls" in a spreadsheet program.
17. Copy the time values from the *Time* column in the "TSS.xls" file to the *Time (hrs)* column in the *XY Series Editor* dialog.
18. Copy the values from the *Concentration* column in the "TSS.xls" file to the *Concentration (mg/L)* column in the *XY Series Editor* dialog.
19. Click **OK**, **Done**, and **OK** to close the three dialogs.
20. Repeat steps 13 to 19 for the upstream and downstream arcs of the main channel, using the data from the following table for the data entered in the *XY Series Editor* instead of opening the spreadsheet.

Time (hours)	Concentration
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 the downstream boundary condition initially. However, the Advection Dispersion module requires that data for constituents is present at all the boundary conditions.

4 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.

5 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 "40m_SS_AD" simulation and select **Export TUFLOW files.** This creates a directory named "TUFLOW" where the files will be written. The directory structure model is the same as that described in the *TUFLOW Users Manual*.
2. Right-click on the "40m_SS_AD" simulation and select **Launch TUFLOW**. This will bring up a console window and launch TUFLOW. This process may take several minutes to complete.

6 Using Log and Check Files

TUFLOW generates several files that can be useful for locating problems in a model. In the directory *data files\TUFLOW \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 doing the following:

1. Select *File* | **View Data file**.
2. Open "40m_SS_AD.tlf" in the *data files\TUFLOW\runs\log* directory.
3. A *View Data File* dialog may appear asking which program to use to open the file. Select "Notepad" or another text editor and click **OK**.
4. Scroll 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 for "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 *data files\TUFLOW\runs\log* directory, there should be a MIF/MID pair of files named "40m_SS_AD_messages.mif".

To view these files:

1. Select *File* | **Open**.
2. Open "40m_SS_AD_messages.mif" to bring up the *Mif/Mid import dialog*. This file contains messages which are tied to the locations where they occur.
3. Under *Read As*, select "GIS layer" from the drop-down menu and click **OK**.
4. Nothing will happen because there are no errors in these files. If the simulation had any errors or warnings, they would show up in this file. Otherwise, the file is empty (as in this case).

It is sometimes difficult to read the messages because they are stacked on top of each other. Use the **Get Attributes** tool to see what the messages are.

To use the **Get Attributes** tool:

5. Click on the object (point at the start of the text string). This will bring up the *Info* 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 **Get Attributes** tool can be used with points, lines, and polygons to check TUFLOW input values.

7 Viewing the Solution

TUFLOW has several kinds of output. All the output data is found in the *data files\TUFLOW\results* folder. Each file begins with the name of the simulation which generated the files.

The results folder contains a 2DM, SUP, MAT, 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** and open the *data files\TUFLOW\results* folder.
2. Select "40m_SS_AD.xmdf.sup" and click the **Open** button. The TUFLOW output is read into SMS in the form of a two-dimensional mesh.
3. If a dialog pops up and asking to replace existing material definitions, click **No**.
4. If a dialog pops up and asks for time units, select "hours".
5. From the Project Explorer, turn off all Map Data, Scatter Data, and Cartesian Grid Data. Turn on and highlight the Mesh Data.
6. Click the **Display Options** tool to open the *Display Options* dialog. From the *2D Mesh* tab, turn on *Contours* and *Vectors*.
7. Switch to the *Contours* tab and select "Color Fill" as the *Contour method*.
8. Click **OK** to close the *Display Options* dialog.

The mesh will be contoured according to the selected dataset and time step. At this point any of the techniques demonstrated in the *Data Visualization* tutorial can be used to visualize the TUFLOW results including film loops and observation plots.

8 Comparing TUFLOW AD and RMA4 Solutions

RMA4 is part of the TABS-MD suite of programs and is used for tracking constituent flow in a 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".

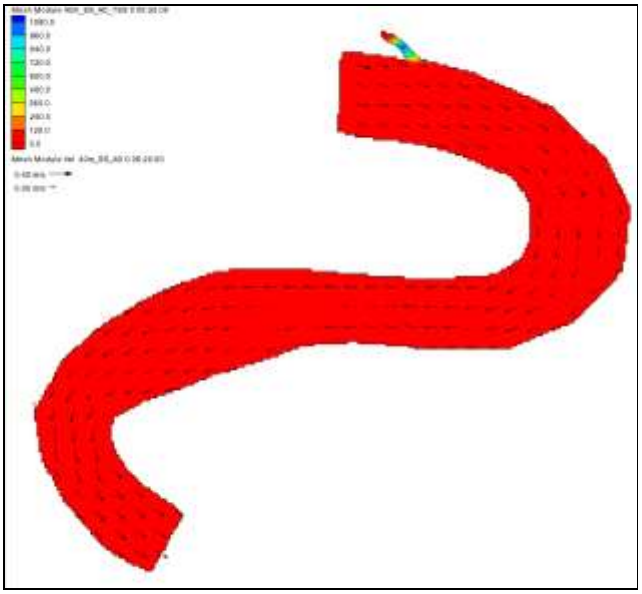


Figure 3 Results obtained in TUFLOW AD

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.

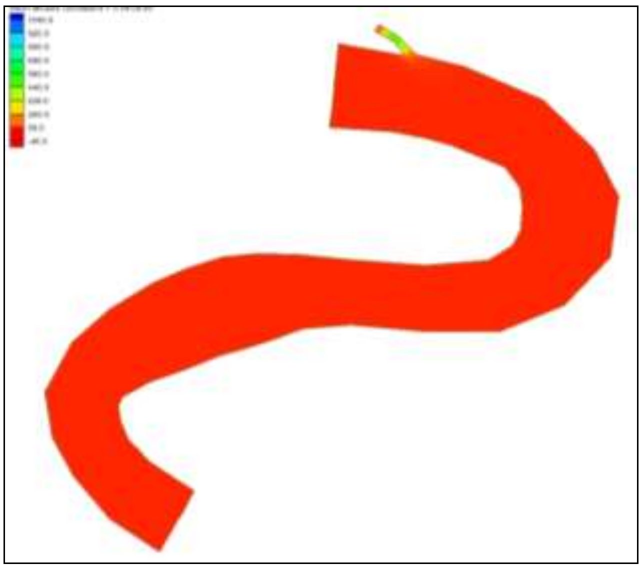


Figure 4 Results obtained in RMA4

The results obtained when running TUFLOW AD are not completely the same as those obtained when running RMA4, even though the same original data were used. The two models compute differently and some differences should be expected.

9 Conclusion

This concludes the TUFLOW AD tutorial. The user may continue to experiment with the SMS interface, or quit the program.