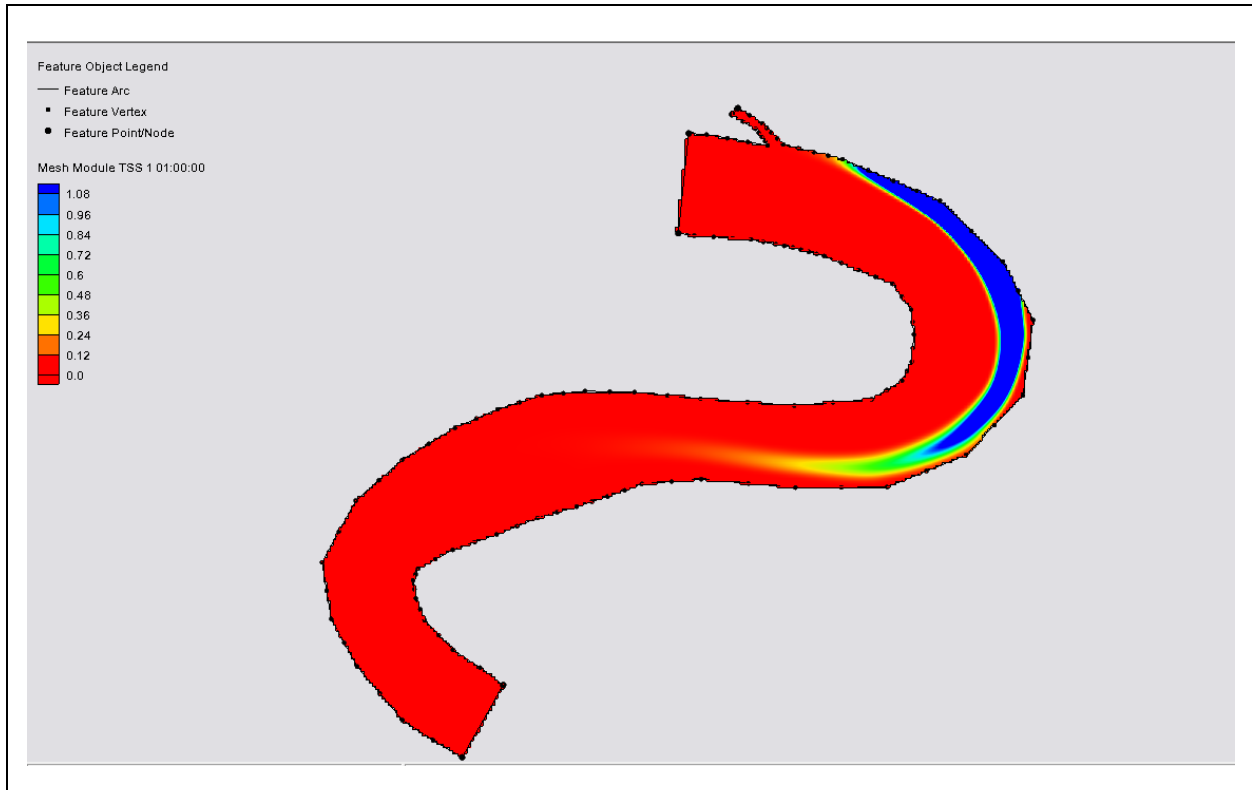


SMS 13.0 Tutorial

TUFLOW – Advection Dispersion



Objectives

Learn to generate a TUFLOW Advection Dispersion project using the SMS interface.

Prerequisites

- TUFLOW 2D
- Overview Tutorial

Requirements

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

Time

- 45–60 minutes

AQUAVEO™



1	Introduction.....	2
2	Getting Started.....	2
2.1	Bathymetry Data	2
2.2	Modifying the Display	3
3	Creating the 2D Model Inputs	4
3.1	TUFLOW Grid.....	4
3.2	Material Properties	5
3.3	2D BC Coverage	6
4	TUFLOW Simulation	8
4.1	Geometry Components.....	8
4.2	Material Sets	9
4.3	Simulation Setup and Model Parameters	9
5	Saving a Project File	11
6	Running TUFLOW	11
7	Using Log and Check Files	11
8	Viewing the Solution	13
9	Comparing TUFLOW AD and RMA4 Solutions	13
10	Conclusion	14

1 Introduction

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

More information about TUFLOW can be obtained from the TUFLOW website¹.

2 Getting Started

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

- Geographic (location) and topographic (elevation) data. Note that all units in TUFLOW are metric.
- Land use data (may be extracted from images or imported as a map file).
- Boundary conditions.

2.1 Bathymetry Data

Start by importing the project. Topographic data in SMS is managed in the scatter module as scatter point datasets or triangulated irregular networks (TINs). SMS uses this data as the source for elevation data in the study area.

To open the scatter point data:

1. Select **File / Open...** to bring up the *Open* dialog.
2. Browse to the *data files* folder for this tutorial and select “madora_ad.sms”.
3. Click **Open** to import the project and exit the *Open* dialog.

¹ See <http://www.tuflow.com/>.

The project should appear similar to Figure 1. It has a scatter point dataset and two coverages: “BC” (boundary conditions) and “materials”. The projection in this project is already set.

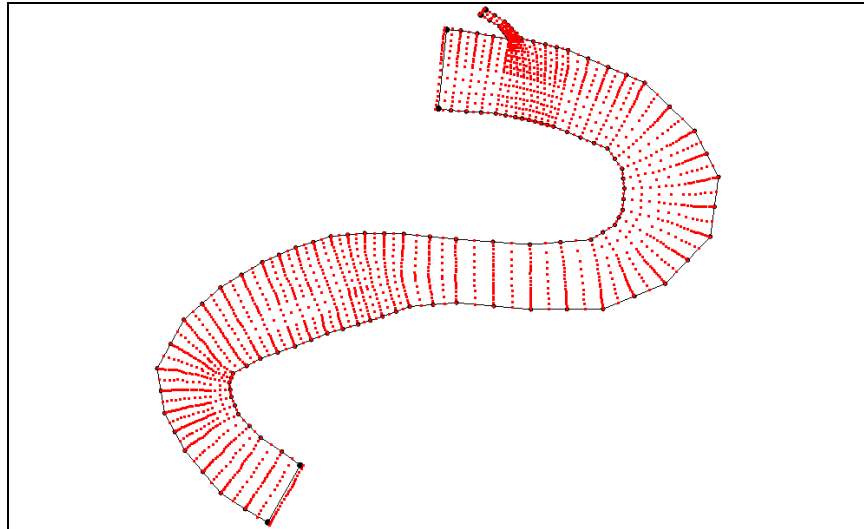


Figure 1 Scatter dataset

2.2 Modifying the Display

Now that the initial data is imported, adjust the display settings by doing the following:

1. Select *Display / Display Options...* to bring up the *Display Options* dialog.
2. Select “Scatter” from the list on the left.
3. On the *Scatter* tab, turn off *Points*.
4. Turn on *Boundary* and *Contours*.
5. On the *Contour* tab, in the *Contour method* section, select “Color Fill” from the first drop-down.
6. Enter “50” as the *Transparency*.
7. Click **OK** to close the *Display Options* dialog.

The Graphics Window should appear similar to Figure 2.

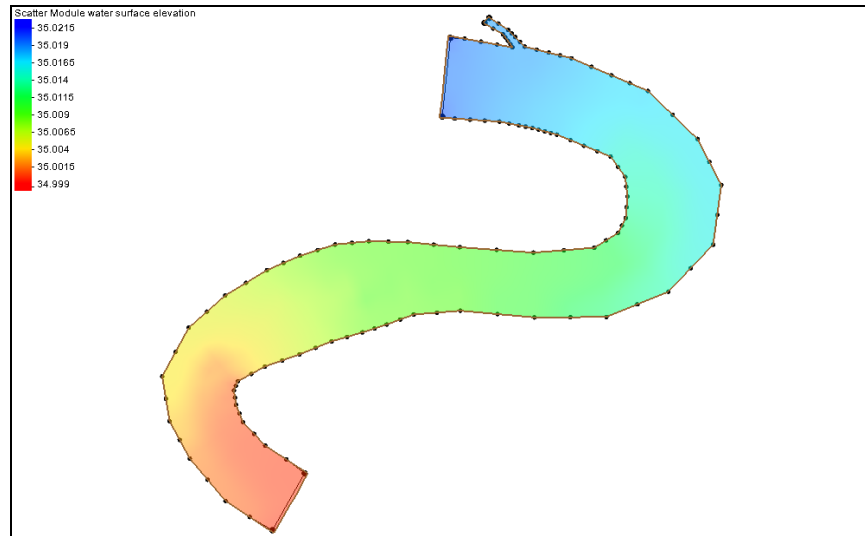





Figure 2 Scatter data with contours

3 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.

3.1 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** to bring up the *New Coverage* dialog.
2. Select *Models | TUFLOW | 2D Grid Extents* as the *Coverage Type*.
3. Enter “TUFLOW Grid” as the *Coverage Name*.
4. Click **OK** to close the *New Coverage* dialog and create the new coverage.
5. Select “ 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 3.
7. Select *Feature Objects | Map → 2D Grid* to bring up the *Map → 2D Grid* dialog.
8. Enter “40m_AD” as the *Grid name*.
9. In the *I Cell Options* section, enter “40.0” as the *Cell Size*.

Notice that this also sets the cell size in the *J Cell Options* section.

10. In the *Elevation Options* section, select “Scatter Set” from the *Source* drop-down and click **Select...** to bring up the *Interpolation* dialog.



11. In the *Interpolation Options* section, select “Single Value” from the *Extrapolation* drop-down.

12. Enter “35.0” as the *Single Value*.

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.

13. Click **OK** to close the *Interpolation* dialog.

14. Select **OK** to close the *Map → 2D Grid* dialog.

This creates a new item named “ 40m_AD” in the Project Explorer under “ Cartesian Grid Data”.

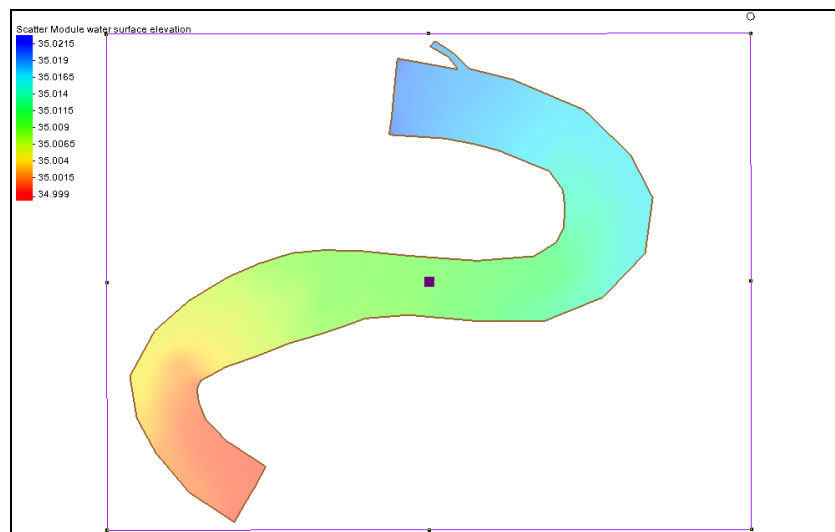


Figure 3 Grid frame enclosing the S-curve

3.2 Material 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 importing the data from MapInfo MIF/MID files. TUFLOW can import the area property data from either GIS data or data mapped to the grid.


In this tutorial, the materials data has been included with the project file. To review the material data, do the following:

1. Select *Edit | Materials Data* to bring up the *Materials Data* dialog.

The listed materials should include bank, channel, sandbar, and overbank materials.

2. Click **OK** to close the *Materials Data* dialog.

3. Turn on “ materials” in the Project Explorer, and select it to make it active.

4. Using the **Select Feature Polygon**  tool, double-click on the polygon at the top of the S-curve (between the small tributary and the inflow boundary) to bring up the *Land Polygon Attributes* dialog.

5. In the *Polygon Type* section, make sure that *Material* is selected and that “bank” is selected from the drop-down.

6. Click **OK** to close the *Land Polygon Attributes* dialog.
7. Repeat steps 4–6 for each polygon, making sure that each polygon is assigned the appropriate material property as indicated in Figure 4.

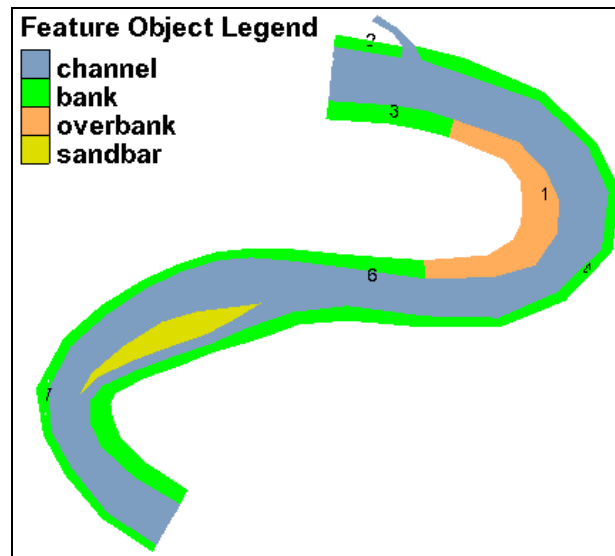


Figure 4 Material properties for each polygons




3.3 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 “ BC” and select *Type | Models | TUFLOW | 1D-2D BCs and Links*.
2. Using the **Select Feature Arc**  tool, double-click on the upstream boundary arc to bring up the *Boundary Conditions* dialog.
3. Select “Flow vs Time (QT)” from the *Type* drop-down.
4. In the *Events* section, click **Edit Events...** to open the *TUFLOW BC Events* dialog.
5. In the *Events* section, click **Add**  if “SS” is not in the *Events* window. This adds an entry for “new_event” to the list.
6. Double-click on “new_event” and enter “SS” as the new name.


7. Click **OK** to close the *TUFLOW BC Events* dialog.
8. In the *Events* section, select “SS” from the list and click **Curve undefined** to bring up the *XY Series Editor* dialog.
9. Outside of SMS, browse to the *data files* folder for this tutorial and open “bc_tss.xls” in a spreadsheet program.
10. In the *Boundary Condition 1* section of the spreadsheet, copy the time values from the *Time* column to the *Time (hrs)* column in the *XY Series Editor* dialog.
11. Copy the flow values from the *Flow* column in the spreadsheet to the *Flow (cms)* column in the *XY Series Editor* dialog.
12. Click **OK** to close the *XY Series Editor* dialog.
13. Click **OK** to close up the *Boundary Conditions* dialog.

To assign boundary conditions for the 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. Select “Flow vs Time (QT)” from the *Type* drop-down.
3. In the *Events* section, select “SS” from the list and click **Curve undefined** to bring up the *XY Series Editor* dialog.
4. From the *Boundary Condition 2* section in the “bc_tss.xls” file, copy the time values from the *Time* column to the *Time (hrs)* column in the *XY Series Editor* dialog.
5. Copy the flow values from the *Flow* column in the spreadsheet to the *Flow (cms)* column in the *XY Series Editor* dialog.
6. Click **OK** to close the *XY Series Editor* dialog.
7. Click **OK** to close up the *Boundary Conditions* dialog.

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. Select “Wse vs Time (HT)” from the *Type* drop-down.
3. In the *Events* section, select “SS” from the list and click **Curve undefined** to bring up the *XY Series Editor* dialog.
4. From the *Boundary Condition 3* section in the “bc_tss.xls” file, copy the time values from the *Time* column to the *Time (hrs)* column in the *XY Series Editor* dialog.
5. Copy the flow values from the *Wse* column in the spreadsheet to the *Wse (m)* column in the *XY Series Editor* dialog.
6. Click **OK** to close the *XY Series Editor* dialog.
7. Click **OK** to close up the *Boundary Conditions* dialog.
8. Select “ BC” to make it active.



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. In the Options section, turn on *Set cell code*.
12. Select *Active* from the *Set cell code* drop-down.
13. Click **OK** to close the *Boundary Conditions* dialog.


4 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 with each including 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 select *New Simulation / TUFLOW*.

A new simulation titled “ Sim” should appear under the “ Simulations” folder in the Project Explorer.

2. Right-click on “ Sim” and select **Rename**.
3. Enter “40m_SS_AD” and press *Enter* to set the new name.







4.1 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 generally 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 “ Components” and select **New 2D Geometry Component**.
2. Right-click on “ 2D Geom Component” and select **Rename**.
3. Enter “40m_geo” and press *Enter* to set the new name.
4. Drag “ 40m_AD”, “ materials”, and “ BC” under “ 40m_geo”.

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:



5. Right-click on “ 40m_geo” and select **Grid Options...** to bring up the *Grid Options* dialog.
6. In the *Materials* section, select *Specify using area property coverage(s)*.
7. Select “bank” from the *Default material* drop-down.
8. In the *Cell codes* section, select *Specify using BC coverage(s)*.
9. Select “Inactive cell – not in mesh” from the *Default code* drop-down.
10. Click **OK** to close the *Grid Options* dialog.

4.2 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 “ Material Sets” and select **New Material Set**.

A new “ Material Set” will appear below the “ Material Sets” folder.

2. Right-click on the new “ Material Set” and select **Rename**.
3. Enter “Madora” and press *Enter* to set the new name.
4. Right-click on “ Madora” and select **Properties** to open the *TUFLOW Material Properties* dialog.
5. Using the table below, enter the values for Mannings n by selecting each material from the list on the left and entering the appropriate value from the table.

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

6. Click **OK** to close the *TUFLOW Material Properties* dialog.


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 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 “ 40m_geo” onto “ 40m_SS_AD” in the Project Explorer.



This creates a link to the geometry component from the simulation.

2. Right-click on “ 40m_SS_AD” and select **2D Model Control...** to open the *TUFLOW 2D Model Control* dialog.
3. Select “Time” from the list on the left:
4. Enter “0” as the *Start Time (hrs)*.
5. Enter “25” as the *End Time (hrs)*.
6. Enter “5” as the *Time Step (s)*.
7. Enter “2” as the *Number of iterations*.
8. Select “Output Control” from the list on the left.
9. In the *Map output* section, select “SMS 2dm” from the *Format Type* drop-down.
10. Enter “0” as the *Start Time*.
11. Enter “600” as the *Interval*.
12. In the *Output datasets* section, turn on *Depth*, *Water Level*, *Flow Vectors*, and *Velocity Vectors*.
13. Turn off any other datasets.
14. In the *Screen/log output* section, enter “6” as the *Display interval*.

While TUFLOW is running, it will write status information every 6 time steps.

15. Select *Water Level* from the list on the left.
16. Enter “35.0” as the *Initial water level (m)*.
17. Select *BC* from the list on the left.
18. Select “SS” from the *BC event name* drop-down.

The advection dispersion data needs to be set up as well:

19. Select *Constituents* from the list on the left.
20. Enter “TSS” as the *Name*.
21. Enter “12.6” for *Longitudinal* under *Dispersion Coefficients*.
22. Click **OK** to close the *TUFLOW 2D Model Control* dialog.
23. Select “ BC” to make it active.
24. Using the **Select Feature Arc**  tool, double-click the upstream arc of the small side channel to open the *Boundary Conditions* dialog again.
25. In the *Advection dispersion* section, click **Advection Dispersion BC...** to bring up the *Advection Dispersion* dialog.
26. In the *Constituents* section, select “TSS” and click **Curve undefined** to open the *XY Series Editor* dialog.
27. Outside of SMS, open the “bc_tss.xls” in a spreadsheet program and copy the time values from the *Time* column in the *TSS Side Channel* section to the *Time (hrs)* column in the *XY Series Editor* dialog.

28. Copy the values from the *Concentration* column in the spreadsheet to the *Concentration (mg/L)* column in the *XY Series Editor* dialog.
29. Click **OK** to close the *XY Series Editor* dialog.
30. Click **Done** to close the *Advection Dispersion* dialog.
31. Click **OK** to close the *Boundary Conditions* dialog.
32. Repeat steps 6–13 for the upstream arc of the main channel, using the *TSS Upstream Downstream* section for the data entered in the *XY Series Editor*.
33. Repeat steps 6–13 for the downstream arc of the main channel, using the *TSS Upstream Downstream* section for the data entered in the *XY Series Editor*.

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 TUFLOW AD model 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 As...** to open the *Save As* dialog.
2. Enter “madora_ad_complete.sms” as the *File name*.
3. Click **Save** to save the project under the new name and close the *Save As* dialog.


6 Running TUFLOW

TUFLOW can be launched from inside 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 “ 40m_SS_AD” 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 “ 40m_SS_AD” simulation and select **Launch TUFLOW**.

This will bring up a console window and launches TUFLOW. This process may take several minutes to complete, depending on the speed of the computer being used.

7 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...** to bring up an *Open* dialog.
2. Browse to the *data files\TUFLOW\runs\log* directory and select “40m_SS_AD.tlf”.
3. Click **Open** to exit the *Open* dialog and bring up the *View Data File* dialog. If the *Never ask this again* option has previously been turned on, this dialog will not appear. If this is the case, skip to step 5.
4. Select the desired text editor from the *Open with* drop-down and click **OK** to exit the *View Data File* dialog and open the file in the desired text editor.
5. Scroll to the bottom of the file.

The bottom of this file reports 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 (search for “ERROR” or “WARNING”), and some are found in the “messages.mif” file (discussed below).



6. When done, close the text editor application and return to SMS.


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:


7. Select *File* / **Open** to bring up the *Open* dialog.
8. Select “40m_SS_AD_messages.mif” and click **Open** to exit the *Open* dialog and bring up the *Mif/Mid import* dialog.


This file contains messages which are tied to the locations where they occur.

9. Under *Read As*, select “GIS layer” from the drop-down menu and click **OK** to close the *Mif/Mid import* dialog.
10. A new “ 40m_SS_AD_messages.mif” entry will appear under “ GIS Data” in the Project Explorer.

If the simulation had any errors or warnings, they would show up in this file. Otherwise, the file is empty (as in this case). When error messages or warnings are shown, it is sometimes difficult to read the messages because they are stacked on top of each other. If needed, use the **Get Attributes**  tool to see what the messages are. Otherwise proceed to the next section of the tutorial.

If there are errors, use this tool by doing the following:

11. Using the **Get Attributes**  tool, click on the object (the 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.

8 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 can contain a variety of different files, including 2DM, SUP, MAT, and DAT files. 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. Click **Open**  to bring up an *Open* dialog.
2. Browse to the *data files\TUFLOW\results* folder and select “40m_SS_AD.xmdf.sup”.
3. Click **Open** to exit the Open dialog and import the SUP file.

The TUFLOW output is imported into SMS in the form of a two-dimensional mesh.

4. If asked to replace existing material definitions, click **No**.
5. If asked for time units, select “hours” and click **OK**.
6. In the Project Explorer, turn off “ Map Data”, “ Scatter Data”, and “ Cartesian Grid Data”.
7. Turn on and select “ Mesh Data”.
8. Click **Display Options**  to open the *Display Options* dialog.
9. Select “2D Mesh” from the list on the left.
10. On the *2D Mesh* tab, turn on *Contours* and *Vectors*.
11. Turn off *Nodes* and *Elements*.
12. On the *Contours* tab, in the *Contour method* section, select “Color Fill” from the first drop-down.
13. On the *Vectors* tab, in the *Vector Display Placement and Filter* section, select “on a grid” from the *Display* drop-down.
14. Enter “20” for both *X spacing* and *Y spacing*.
15. 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.

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

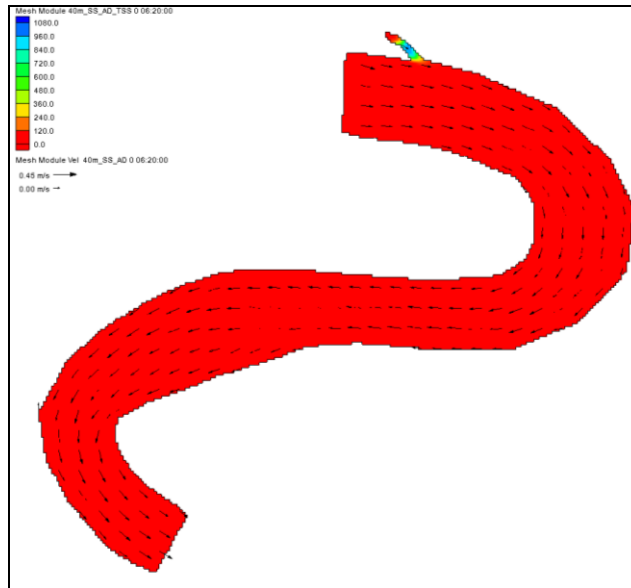


Figure 5 Results obtained in TUFLOW AD

These results were compared to the results obtained in an RMA4 simulation. Figure 6 shows the result obtained in RMA4 at time “04:00:00”. This is about two 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 6 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.

10 Conclusion

This concludes the “TUFLOW Advection Dispersion” tutorial. Continue to experiment with the SMS interface, or quit the program.