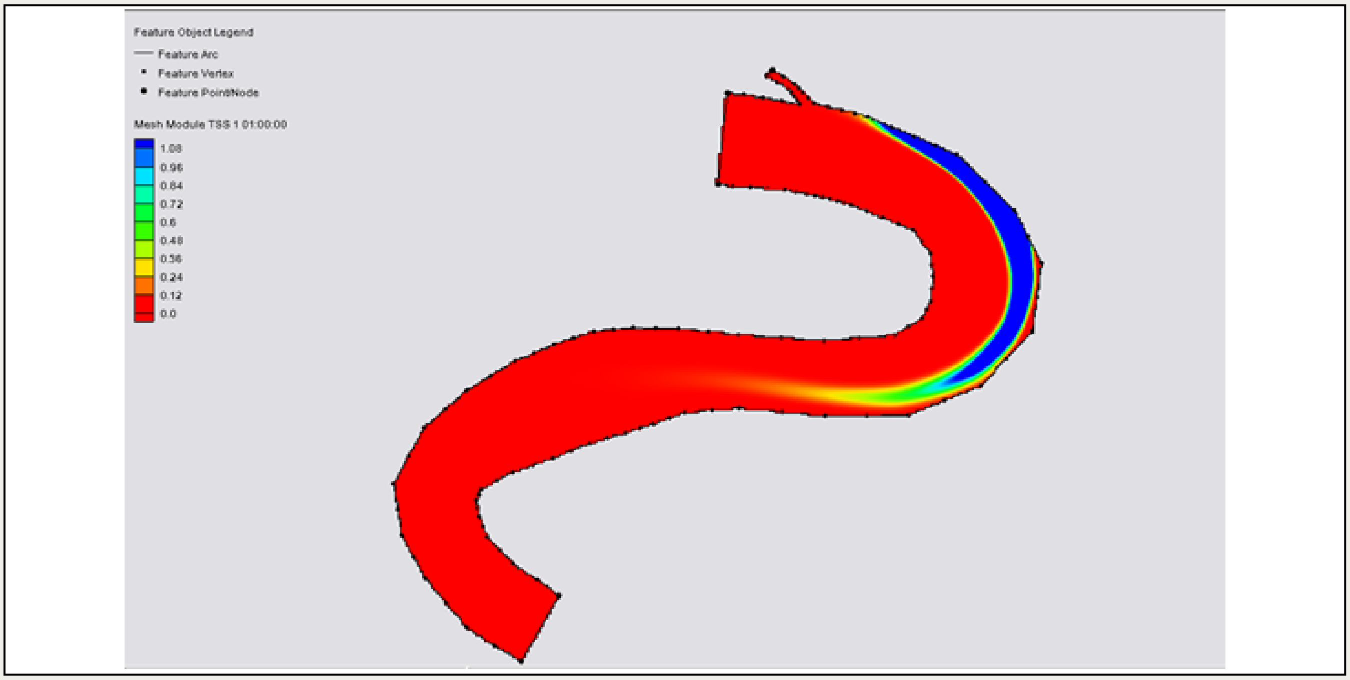
SMS 14.0 Tutorial

TUFLOW – Advection Dispersion

Set up a TUFLOW AD model



Objectives

This tutorial instructs how to generate a TUFLOW Advection Dispersion project using the SMS interface.

Prerequisite Tutorials

- Overview
- TUFLOW

Required Components

- SMS Core
- TUFLOW Model & Interface

Time

- 45–60 minutes

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 website[1].

2 Getting Started

SMS modeling studies require the use of 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 the *File* | **Open…** menu command to bring up the *Open* dialog.
2. Browse to the *data files* folder for this tutorial and select the “madora_ad.sms” file.
3. Click the **Open** button 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.

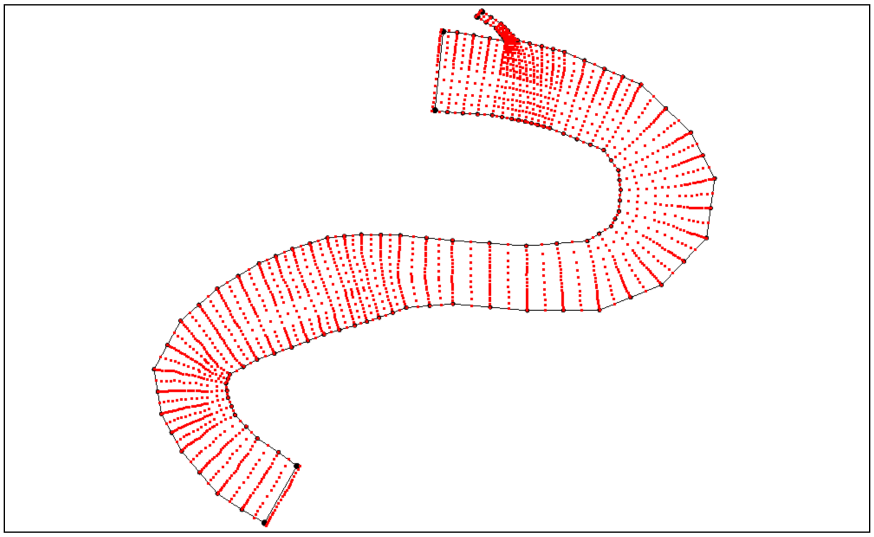


Figure 1 Scatter dataset

2.2 Modifying the Display

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

1. Select the *Display* | **Display Options…** menu command to bring up the *Display Options* dialog.
2. From the list on the left, select "Scatter".
3. On the *Scatter* tab, turn off the *Points* checkbox.
4. Turn on the *Boundary* and *Contours* checkboxes.
5. On the *Contours* tab, in the *Contour method* section, select "Color Fill" from the first drop-down.
6. For the *Transparency,* enter "50".
7. Click the **OK** button to close the *Display Options* dialog.

The Graphics Window should appear similar to Figure 2.

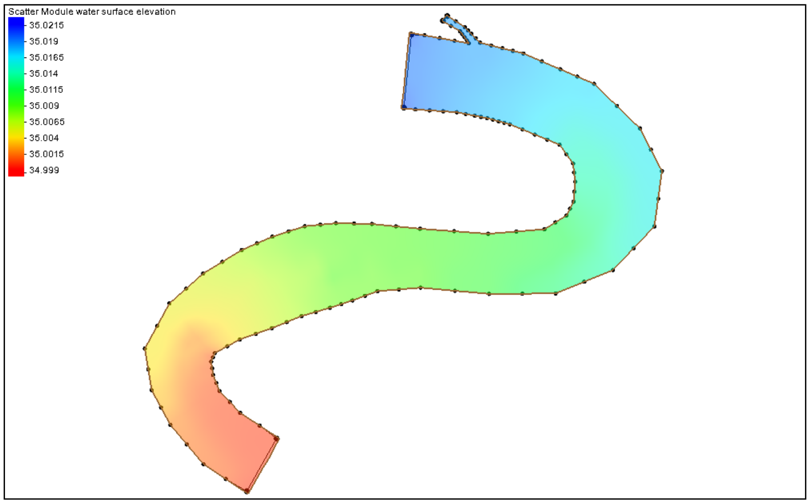


Figure 2 Scatter data with contours

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. In the Project Explorer, right-click on the " Map Data" folder and select the **New Coverage** context menu command to bring up the *New Coverage* dialog.
2. For the *Coverage Type,* select *Models* | *TUFLOW* | **2D Grid Extents**.
3. For the *Coverage Name,* enter "TUFLOW Grid".
4. Click the **OK** button 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** dynamic tool, create a grid frame enclosing the entire S-curve, as shown in Figure 3.

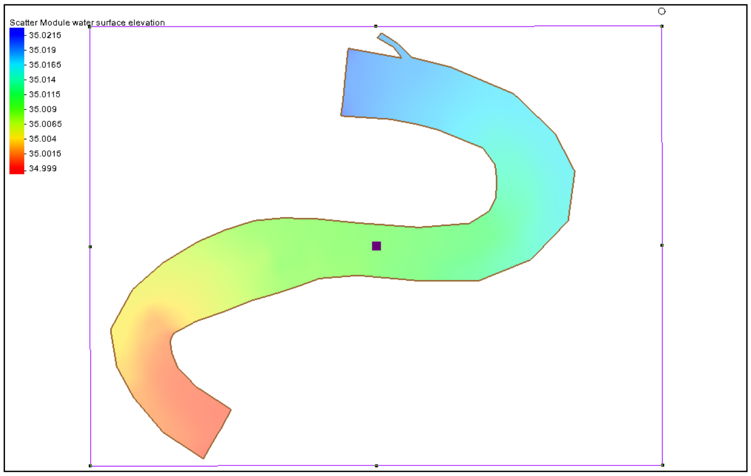


Figure 3 Grid frame enclosing the S-curve

7. Select the *Feature Objects* | **Map** → **2D Grid** menu command to bring up the *Map → 2D Grid* dialog.
8. For the *Grid name,* enter "40m_AD".
9. In the *I Cell Options* section, in C*ell Size,* enter "40.0" (m).

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

10. In the *Elevation options* section, from the *Source* drop-down select "Scatter Set".
11. Click the **Select…** button to bring up the *Interpolation* dialog.
12. In the *Interpolation Options* section from the *Extrapolation* drop-down, select "Single Value".
13. For the *Single Value,* enter "35.0".

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.

14. Click the **OK** button to close the *Interpolation* dialog.
15. Click the **OK** button to close the *Map → 2D Grid* dialog.

This creates a new 2D grid item named " 40m_AD" under " Cartesian Grid Data" in the Project Explorer.

3.2 Material Properties

Area property coverages define 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. Moreover, SMS 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 channel, bank, sandbar, and overbank materials.

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

3. In the Project Explorer, turn off the " 40m_AD" grid data checkbox, to make the next steps easily accessible.
4. In the Project Explorer, turn on the " materials" map coverage checkbox, and select it to make it active.
5. Using the **Select Feature Polygon** dynamic tool, double-click on the polygon at the top of the S-curve (between the small tributary and the inflow boundary, labeled 2 in Figure 4) to bring up the *Land Polygon Attributes* dialog.
6. In the *Polygon Type* section, select the *Material* checkbox and from the drop-down select "bank".
7. Click the **OK** button to close the *Land Polygon Attributes* dialog.
8. Repeat steps 4–6 for each polygon, making sure that each polygon is assigned the appropriate material property as indicated in Figure 4.

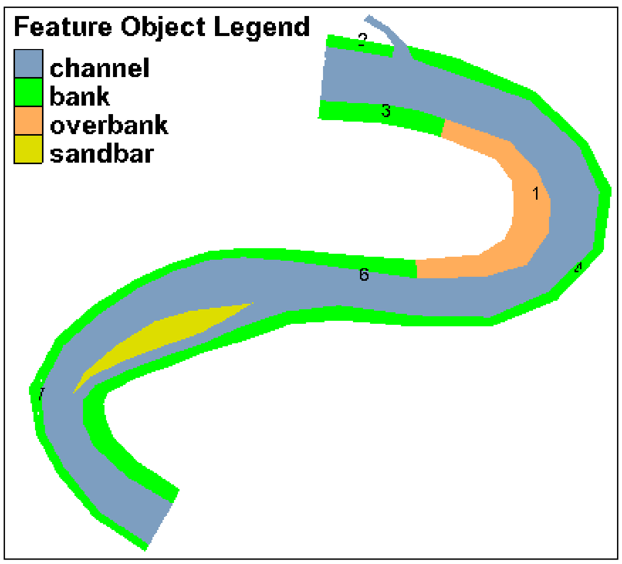


Figure 4 Material properties for each polygon

3.3 Assign Main Upstream 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. In the Project Explorer, right-click on the " BC" map coverage and select the *Type | Models | TUFLOW |* **1D-2D BCs and Links** context menu item.

2. Using the **Select Feature Arc** dynamic tool, double-click on the upstream boundary arc (running across the top arc of main channel) to bring up the *Boundary Conditions* dialog.
3. From the *Type* drop-down, select “Flow vs Time (QT)”.
4. In the *Events* section, click the **Edit Events…** button to open the *TUFLOW BC Events* dialog.
5. If “SS” is not in the *Events* window, click the **Add** button. This adds an entry for “new_event” to the list.
6. Double-click on “new_event” and as the new name, enter “SS”.
7. Click the **OK** button to close the *TUFLOW BC Events* dialog.
8. In the *Events* section, select “SS” from the list.
9. Click the **Curve undefined** button to bring up the *XY Series Editor* dialog.
10. Outside of SMS, browse to the *data_files* folder for this tutorial and open “bc_tss.xls” in a spreadsheet program.
11. In the *Boundary Condition 1* section of the spreadsheet in the *Time* column, copy the time and flow values (cells A3 through B6).
12. Paste the time and flow values from the spreadsheet into the first cell of the *Time (hrs)* column in the *XY Series Editor* dialog. This will fill the table and update the graph on the right.
13. Click the **OK** button to close the *XY Series Editor* dialog.
14. Click the **OK** button to close the *Boundary Conditions* dialog.

3.4 Assign Side Channel 2D BC Coverage

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. From the *Type* drop-down, select “Flow vs Time (QT)”.
3. In the *Events* section, select “SS” from the list.
4. Click the **Curve undefined** button to bring up the *XY Series Editor* dialog.
5. In the outside spreadsheet from the *Boundary Condition 2* section, in the *Time* column, copy the time and flow values (cells D3 through E6).
6. Paste the values into the first cell in the *Time (hrs)* column in the *XY Series Editor* dialog. This will fill the table and update the graph on the right.
7. Click the **OK** button to close the *XY Series Editor* dialog.
8. Click the **OK** button to close the *Boundary Conditions* dialog.

3.5 Assign Main Downstream 2D BC Coverage

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 (running across the lower arc of main channel) to bring up the *Boundary Conditions* dialog.

2. From the *Type* drop-down, select “Wse vs Time (HT)”.
3. In the *Events* section, select “SS” from the list and click the **Curve undefined** button to bring up the *XY Series Editor* dialog.
4. In the outside spreadsheet from the *Boundary Condition 3* section, in the *Time* column, copy the time and flow values (cells G3 through H6).
5. Paste the values into the first cell of the *Time (hrs)* column in the *XY Series Editor* dialog. This will fill the table and update the graph on the right.
6. Click the **OK** button to close the *XY Series Editor* dialog.
7. Click the **OK** button to close the *Boundary Conditions* dialog.
8. Select the “ BC” map coverage to make it active.
9. Select the *Feature Objects* | **Build Polygons** menu command.
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 the *Set cell code* checkbox.
12. From the *Set cell code* drop-down, select *Active*.
13. Click the **OK** button 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. In an empty part of the Project Explorer, right-click and select the *New Simulation* | **TUFLOW** context menu command.

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

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

4.1 Assign 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 the " Components" folder and select the **New 2D Geometry Component** context menu command.
2. Right-click on the " 2D Geom Component" component and select the **Rename** context menu command.
3. Enter "40m_geo" and press *Enter* to set the new name.
4. Drag the " 40m_AD", " materials", and " BC" coverages under the " 40m_geo" component.

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 the " 40m_geo" component and select the **Grid Options…** context menu command to bring up the *Grid Options* dialog.
6. In the *Materials* section, select the *Specify using area property coverage(s)* radio button.
7. From the *Default material* drop-down, select "bank".
8. In the *Cell codes* section, select the *Specify using BC coverage(s)* radio button.
9. From the *Default code* drop-down, select "Inactive cell – not in mesh".
10. Click the **OK** button to close the *Grid Options* dialog.

4.2 Define 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 the **New Material Set** context menu command.

A new " Material Set (2)" material set will appear below the " Material Sets" folder.

2. Right-click on the new " Material Set (2)" material set and select **Rename**.
3. Enter "Madora" and press *Enter* to set the new name.
4. Right-click on the " Madora" material set and select the **Properties** context menu command to open the *TUFLOW Material Properties* dialog.
5. Using the table below, enter the values for Manning's *n* by selecting each material from the list on the left and entering the appropriate value from the table.

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

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

4.3 Simulation Setup and Model Parameters

The simulation includes a link to the geometry component and links 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. In the Project Explorer, drag the " 40m_geo" component under the " 40m_SS_AD" simulation.

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

2. Right-click on " 40m_SS_AD" and select the **2D Model Control…** context menu command to open the *TUFLOW 2D Model Control* dialog.

Set the Time Options

1. From the list on the left, select "Time".
2. For the *Start time (hrs)* enter "0".
3. For the *End time (hrs),* enter "25".
4. For the *Timestep (s),* enter "5".
5. For the Number of iterations, enter "2".

Set the Output Control Options

1. From the list on the left, select "Output Control".
2. In the *Map output* section, from the *Format type* drop-down, select "SMS 2dm".
3. For the *Start Time,* enter "0" (hrs).
4. For the *Interval,* enter "600" (s).
5. In the Output datasets section, turn on Depth, Water Level, Flow Vectors, and Velocity Vectors.
6. Turn off any other datasets.
7. In the *Screen/log output* section, for the *Display interval,* enter "6" (timesteps).

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

Set the Water Level Options

1. From the list on the left, select *Water Level*.
2. For the *Initial water level (m),* enter "35.0".
3. Turn on the Override default instability level (10m above highest elevation) checkbox
4. For the Instability level (m), enter "40".

Set the Boundary Conditions Options

1. From the list on the left, select *BC*.
2. From the *BC event name* drop-down, select "SS".

Set the Constituents Options

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

1. From the list on the left, select *Constituents*.
2. For the *Name* enter "TSS".
3. For *Longitudinal* enter "12.6".
4. Click the **OK** button to close the *TUFLOW 2D Model Control* dialog.

Assigning Side Channel Boundary Conditions

1. Select the " BC" map coverage to make it active.
2. Using the **Select Feature Arc** tool, double-click the upstream arc of the small side channel to open the *Boundary Conditions* dialog again.
3. From the *Type* drop-down, select "Flow vs Time (QT)".
4. In the *Advection dispersion* section, click the **Advection Dispersion BC…** button to bring up the *Advection Dispersion* dialog.
5. In the *Constituents* section, select "TSS".
6. Click the **Curve undefined** button to open the *XY Series Editor* dialog.
7. In the "bc_tss.xls" spreadsheet from the *TSS Side Channel* section, in the *Time* column, copy the time values.
8. Paste the time values into the *Time (hours)* column in the *XY Series Editor* dialog.
9. In the spreadsheet from the *TSS Side Channel* section, in the *Concentration* column, copy the values.
10. Paste the values into the *Concentration (mg/L)* column in the *XY Series Editor* dialog.
11. Click the **OK** button to close the *XY Series Editor* dialog.
12. Click the **Done** button to close the *Advection Dispersion* dialog.
13. Click the **OK** button to close the *Boundary Conditions* dialog.
14. Repeat steps 2–13 for the upstream arc of the main channel, using the *TSS Upstream Downstream* section for the data entered in the *XY Series Editor*.
15. Repeat steps 2–13 for the downstream arc of the main channel, using the *TSS Upstream Downstream* section for the data entered in the *XY Series Editor*.

It can be seen that 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 the *File* | **Save As…** menu command to open the *Save As* dialog.
2. For the *File name,* enter "madora_ad_complete.sms".
3. Click the **Save** button 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, confirm that the TUFLOW and TUFLOW AD licenses are active and available to SMS.

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 described in the *TUFLOW User's Manual*.

2. Right-click on " 40m_SS_AD" and select **Launch TUFLOW**.

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

3. When the simulation is complete, click the **OK** button in the *TUFLOW* dialog to close the command windows and return to SMS.

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 any warning or error messages.

Viewing TLF Files

This file can be opened with a text editor by doing the following:

1. Select the *File* | **View Data File…** menu command to bring up an *Open* dialog.
2. Browse to the *data files\TUFLOW\runs\log* directory and select "40m_SS_AD.tlf".
3. Click the **Open** button 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 6.
4. Select the desired text editor from the *Open with* drop-down.
5. Click the **OK** button to exit the *View Data File* dialog and open the file in the desired text editor.
6. Scroll to the bottom of the file.

The bottom of this file reports if the run finished, whether the simulation was stable, and reports 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).

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

Viewing MIF Files

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 the *File* | **Open…** menu command to bring up the *Open* dialog.
2. Select "40m_SS_AD_messages.mif".
3. Click the **Open** button to exit the *Open* dialog and bring up the *Mif/Mid import* dialog.

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

4. Under *Read As*, select "GIS layer" from the drop-down menu.
5. Click the **OK** button to close the *Mif/Mid import* dialog.
6. If prompted to specify a projection, click the **OK** button to accept the default.
7. 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** dynamic 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:

8. Using the **Get Attributes** dynamic tool, click on the object (the point at the start of the text string).

This will bring up the *Info* dialog which shows the attributes (in this case, text) of the object or objects at the selected 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** dynamic 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 that 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 the **Open** macro to bring up an *Open* dialog.
2. Browse to the *data files\TUFLOW\results* folder and select the "40m_SS_AD.xmdf.sup" file.
3. Click the **Open** button 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 prompted to specify a projection, click the **OK** button to accept the default.
5. If prompted to replace existing material definitions, click the **No** button.
6. If prompted for time units, select "hours" and click the **OK** button.
7. In the Project Explorer, turn off the " Map Data", " Scatter Data", and " Cartesian Grid Data" checkboxes.

8. Select the " Mesh Data" folder and turn on the checkbox.
9. Click the **Display Options** macro to open the *Display Options* dialog.
10. From the list on the left, select "2D Mesh".
11. On the *2D Mesh* tab, turn on *Contours* and *Vectors*.
12. Turn off *Nodes* and *Elements*.
13. On the *Contours* tab, in the *Contour method* section, select "Color Fill" from the first drop-down.
14. On the *Vectors* tab, in the *Vector Display Placement and Filter* section, from the *Display* drop-down, select "on a grid".
15. Enter "20" for both *X spacing* and *Y spacing*.
16. Click the **OK** button to close the *Display Options* dialog.
17. In the *Time Step* window, select the 09:00:00 time-step.

The mesh will be contoured according to the selected dataset and time step. Choosing the Vector Velosity mesh data should appear similar to Figure 5. 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.

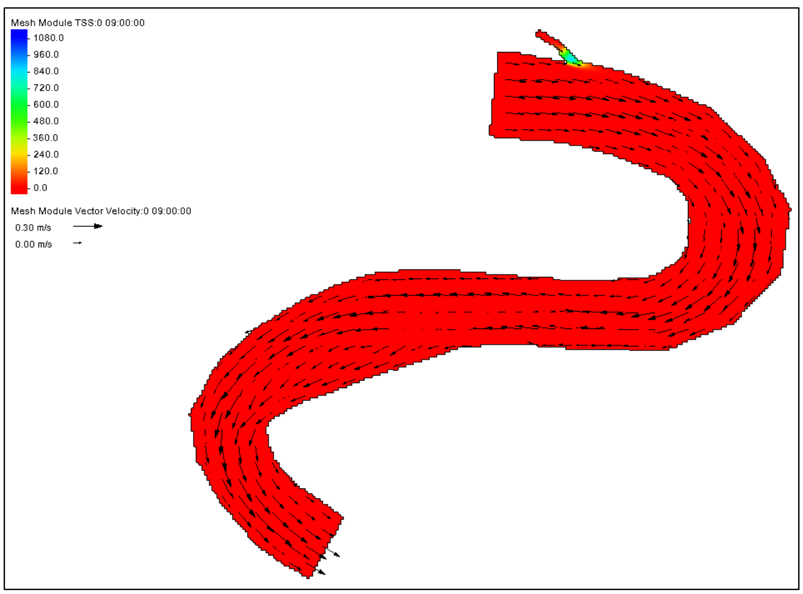


Figure 5 Vector display

9 Conclusion

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

