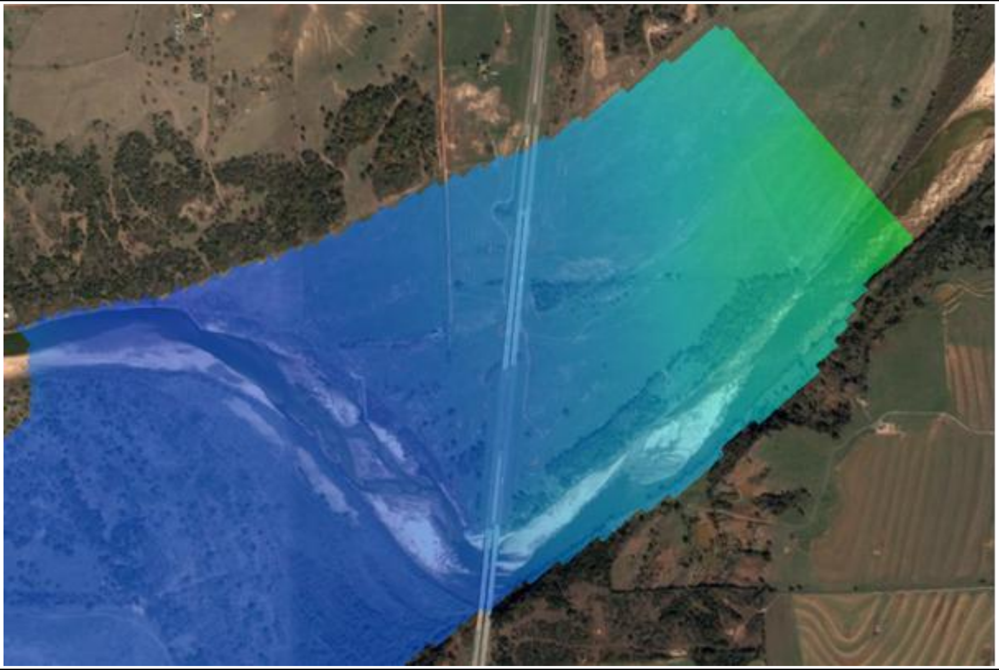
SMS 14.0 Tutorial

TUFLOW 2D Hydrodynamics

Set up a TUFLOW model



Objectives

This tutorial describes the generation of a TUFLOW project using the SMS interface utilizing only the two dimensional flow calculation capabilities of the TUFLOW model. It is suggested that this tutorial be completed before the TUFLOW 1D/2D tutorial.

Prerequisite Tutorials

- Overview

Required Components

- SMS Core
- TUFLOW Model & Interface

Time

- 20–30 minutes

AQUAVEO™

1 Introduction

TUFLOW is a hydraulic model with a wide range of potential applications. It can include 2D only or combined 1D/2D models. The 2D model domains solve the full shallow water equations using the finite difference method. It handles wetting and drying in a very stable manner. More information about TUFLOW can be obtained from the TUFLOW website.[1]

The area used in the tutorial is where I-35 crosses the Cimarron River in Oklahoma, about 50 miles north of Oklahoma City.

1.1 Getting Started

Before starting this tutorial, do the following:

1. Start SMS.
2. If SMS is already open, select the *File* | **New** menu command.
3. If asked to clear all data in SMS, click the **Yes** button.

1.2 Opening an Existing Project

Begin by opening an existing project containing imagery, elevation data, material zone polygons, and a model grid. For more information on generating a grid, see the "Creating a Cell-Centered Grid" tutorial.

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 "start.sms" file.
3. Click the **Open** button to import the project file and exit the *Open* dialog.

The project should appear similar to Figure 1.

[1] See http://www.tuflow.com/ for more details.

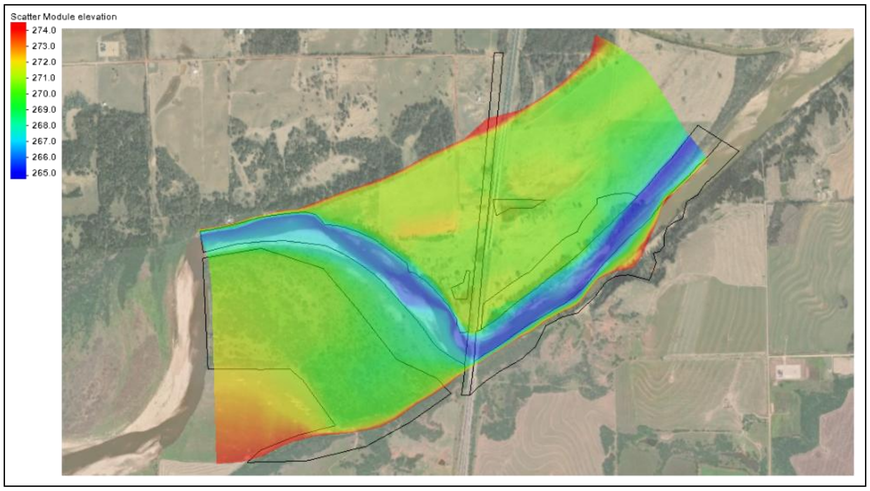


Figure 1 Contours with river portion showing blue

2 Creating the 2D Model Inputs

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

2.1 Area Properties

Area property coverages define the material zones of the grid. This can be created by digitizing directly from an image, or by importing the data from a shapefile. SMS also supports importing the data from MapInfo MIF and MID files.

TUFLOW can read the area property data from either GIS data or data mapped to the grid. In this tutorial, GIS data will be used because it is easy to edit, and generally results in smaller input files and faster runtimes. A shapefile containing the material zone polygons is already loaded in the project. This file will be converted to a coverage in the Map module.

To create the material zone polygons in the Map module:

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. In the *Coverage Type* section, select *Generic* | **Area Property**.
3. For the *Coverage Name,* enter "materials".
4. Click the **OK** button to exit the *New Coverage* dialog and create the new coverage.
5. Select the " materials" coverage to make it active.

When converting GIS data to feature objects, the feature objects are added to the active coverage.

6. In the " GIS Data" folder, select the " materials.shp" shape file.
7. Select the *Mapping* | **Shapes** → **Feature Objects** menu command to bring up the *Step 1 of 2* page of the *GIS to Feature Objects Wizard* dialog.

8. From the *Select a coverage for mapping* drop-down, select “materials”.
9. Click the **Next >** button to go to the *Step 2 of 2* page of the *GIS to Feature Objects Wizard* dialog.
10. In the *MATNAME* column on the *Mapping* row, select “Material” from the drop-down.
11. Click the **Finish** button to close the *GIS to Feature Objects Wizard* dialog.

Notice that the area property coverage contains polygons (Figure 2), but the polygons do not cover the entire domain. Areas not contained inside a polygon will be assigned a default material value. The default material for the simulation is grassland. This material hasn’t been created yet, since it was not part of the area property coverage.

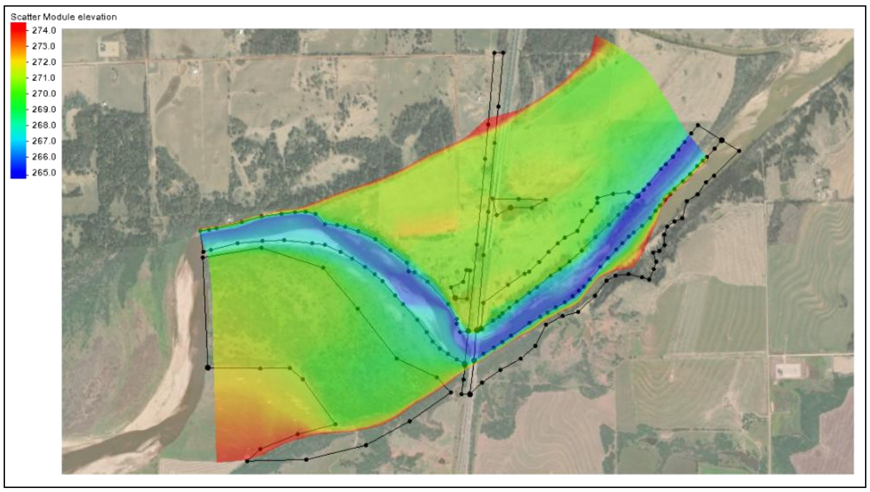


Figure 2 The shapefile polygons do not cover the entire domain

To create this material:

12. Select the *Edit* | **Materials Data** menu command to bring up the *Materials Data* dialog.
13. At the bottom of the *Materials* section, click the **New** button to create a new material called “material 06”.
14. Double-click on the “material 06” name and rename the material to “grasslands”.
15. Click the **OK** button to close the *Materials Data* dialog.

2.2 2D Boundary Condition Coverage

The boundary conditions (BC) for the model need to be specified. This model includes a flow rate boundary condition on the upstream portion of the model, and a water surface elevation boundary condition on the downstream portion of the model. The locations of these boundaries have already been digitized and are included on the BC coverage.

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. Component names must be unique within each project.

To assign the upstream boundary condition:

1. Turn on the " BC" map data checkbox and select the coverage to make it active.
2. Right-click on " BC" and select the *Type* | *Models* | *TUFLOW* | **1D-2D BCs and Links** context menu item.

2.3 Upstream Boundary Condition

Inflow boundary arcs should be created such that constant water surface (head) can be assumed along the arc. The arc as shown in Figure 3 is angled upstream in the floodplain as a better approximation of the correct equal head condition.

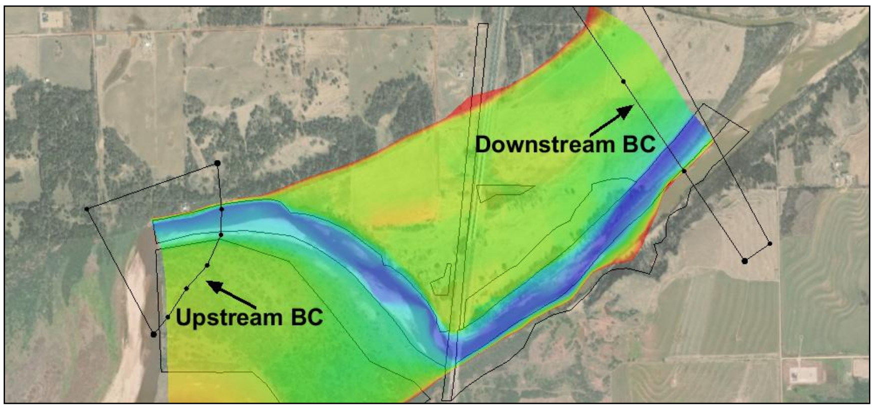


Figure 3 Locations of upstream and downstream BCs

1. Using the **Select Feature Arc** dynamic tool, double-click on the upstream BC arc to bring up the *Boundary Conditions* dialog.
2. From the *Type* drop-down, select "Flow vs Time (QT)".
3. In the *Events* section, click the **Edit Events...** button to open the *TUFLOW BC Events* dialog.
4. In the *Events* section, click the **Add** button to create an event titled "new_event".
5. Double-click on "new_event" and enter "100 year" as the new name.
6. Click the **OK** button to close the *TUFLOW BC Events* dialog.
7. Select "100 year" from the list of events and click the **Curve undefined** button to bring up the *XY Series Editor* dialog.
8. Click the **Import…** button to bring up the *Open File* dialog.
9. Select the "upstream.xys" file and click the **Open** button to import the file and exit the *Open File* dialog.

The values in the *XY Series Editor* dialog can be copied and pasted from a spreadsheet application. After importing the data, the dialog should appear similar to Figure 4.

10. Click the **OK** button to close the *XY Series Editor* dialog.
11. Click the **OK** button to close the *Boundary Conditions* dialog.

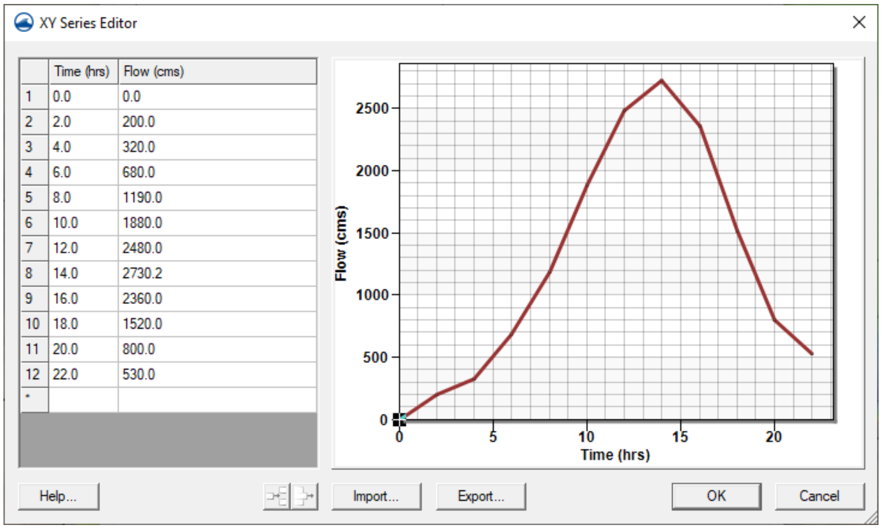


Figure 4 XY Series Editor dialog showing the curve

2.4 Downstream Boundary Condition

To assign the downstream boundary condition:

1. Using the **Select Feature Arc** dynamic tool, double-click the downstream boundary condition arc to bring up the *Boundary Conditions* dialog.
2. From the *Type* drop-down, select “Wse vs Time (HT)”.
3. In the *Events* section, from the list of events, select “100 year”.
4. Click the **Curve undefined** button to bring up the *XY Series Editor* dialog.
5. Click the **Import…** button to bring up the *Open File* dialog.
6. Select the “downstream.xys” file and click the **Open** button to import the data and exit the *Open File* dialog.

The dialog should appear similar to Figure 5.

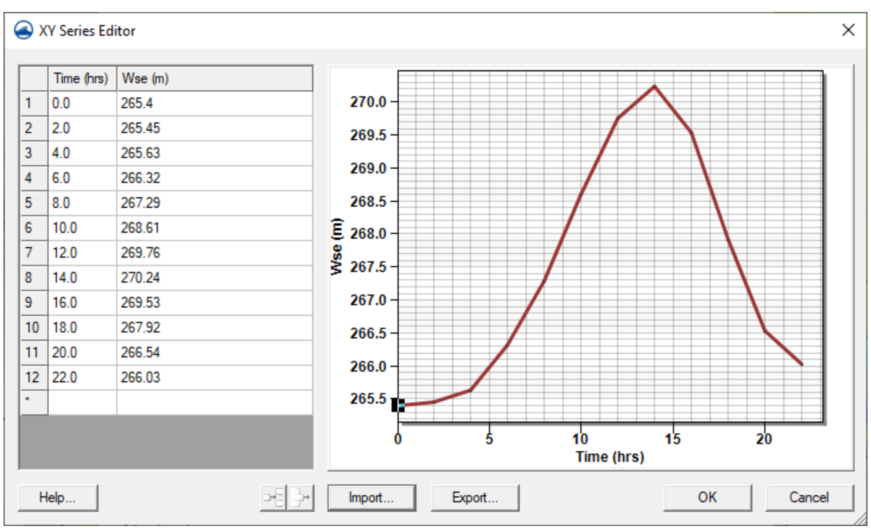


Figure 5 XY Series editor for the downstream BC

The values from the XYS file should now be listed in both columns, and the graph on the right will show a steep curve (Figure 5).

7. Click the **OK** button to close the *XY Series Editor* dialog.
8. Click the **OK** button to close the *Boundary Conditions* dialog.

2.5 Grid Activity Zones

In this simulation, the grid uses cell codes (active/inactive) based upon boundary condition coverages. The default is for all the cells to be active. It is necessary to turn off all the cells upstream of the inflow boundary condition and downstream of the water surface boundary condition (Figure 6).

This can be done using polygons in the boundary condition coverage and setting their attributes to be inactive code polygons. TUFLOW will use code polygons to deactivate the grid cells contained by the null polygons.

To assign boundary conditions to the polygons:

1. Using the **Select Feature Polygon** dynamic tool, double-click on the downstream polygon to bring up the *Boundary Conditions* dialog.
2. In the *Options* section, turn on *Set cell code* and from the drop-down select “Inactive – not in mesh”.
3. Click the **OK** button to close the *Boundary Conditions* dialog.
4. Repeat steps 1–3 for the upstream polygon.
5. In the Graphics Window, click anywhere outside of the polygons to deselect all polygons.

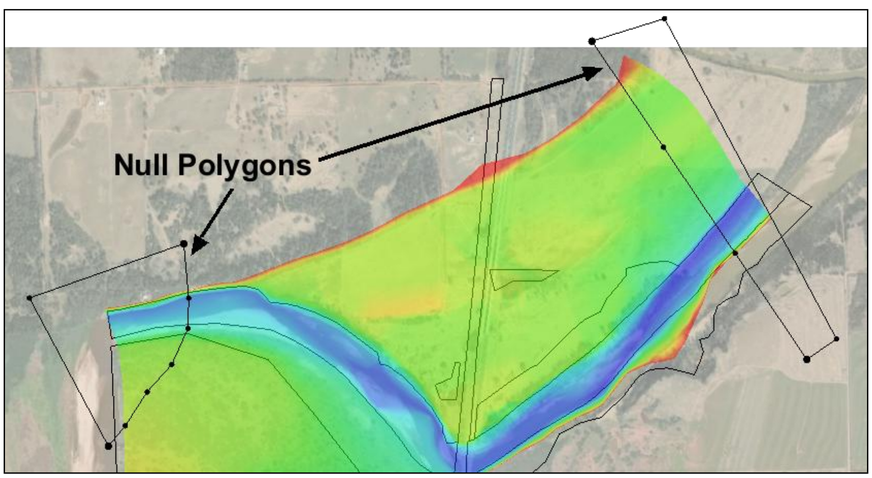


Figure 6 Null polygons

3 TUFLOW Simulation

As mentioned earlier, a TUFLOW simulation is comprised of a grid, feature coverages, and model parameters. SMS allows for the creation of multiple simulations, each including links to these items.

A link is like a shortcut in Windows: the data is not duplicated, but SMS knows where to go to get the required data. The use of links allows these items to be shared between multiple simulations without increasing the size of the project file. A simulation also stores the model parameters used by TUFLOW.

To create the TUFLOW simulation:

1. In an empty space at the bottom of the Project Explorer, right-click and select the *New Simulation* | **TUFLOW** context menu command.

This creates the TUFLOW folder and several new folders used throughout the rest of the tutorial. Under the Project Explorer item “ Simulations” is an item named “ Sim”.

2. Right-click on the “ Sim” simulation and select the **Rename** context menu command.
3. Enter “100year_20m” and press *Enter* to set the new name.

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: 2D boundary condition coverages (if they include code polygons), geometry modification coverages, 2D spatial attribute coverages, and area property coverages.

To create and set up 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” geometry component and select the **Rename** context menu command.
3. Enter “20m_geo” and press *Enter* to set the new name.
4. Drag the “ materials” and “ BC” map coverages, and the “ 20m” grid data under the “ 20m_geo” geometry component.

Notice that the icon for the geometry component is now “ 20m_geo”. This indicates a grid is part of the component.

Because an area property coverage and a default material exist, they need to be associated with the grid. This is specified in the *Grid Options* dialog. At the same time, it will be specified that the grid will use cell-codes from boundary conditions coverages.

To do this:

5. Right-click on “ 20m_geo” 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 “grasslands”.
8. In the *Cell codes* section, select the “Specify using BC coverage(s)” radio button.
9. From the *Default code* drop-down, select “Water cell”.
10. Click the **OK** button to exit the *Grid Options* dialog.

3.2 Material Sets

Now that a simulation has been created, the material properties need to be defined. There is already a " Material Sets" folder, but 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 to create a new " Material Set".
2. Right-click on " Material Set" and select the **Properties** context menu command to bring up the *TUFLOW Material Properties* dialog.
3. From the list on the left, select "channel".
4. On the right, as the *n* value, enter "0.03".

This is the Manning's *n* value.

5. Repeat steps 3–4 for each of the remaining materials in the table below.

Material	Manning's *n*
forest	0.1
light forest	0.08
channel	0.03
Roadway	0.02
grasslands	0.06

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

3.3 Simulation Setup and model parameters

The simulation includes a link to the geometry component, as well as 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. A model could be created where this is not the case.

The TUFLOW model parameters include timing controls, output controls, and various model parameters. To create the link to the geometry component and set up the model control parameters, do the following:

1. Drag the " 20m_geo" component under the " 100year_20m" simulation in the Project Explorer.
2. Right-click on " 100year_20m" and select the **2D Model Control…** context menu command to open the *TUFLOW 2D Model Control* dialog.
3. From the list on the left, select "Output Control".
4. In the *Map output* section, from the *Format type* drop-down, select "SMS 2dm".
5. For the *Start Time,* enter "0" (hrs).
6. For the *Interval,* enter "900" (s).
7. In the *Output datasets* section, turn on *Depth*, *Water Level*, *Flow Vectors*, and *Velocity Vectors*.
8. 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.

9. From the list on the left, select "Time".
10. For the *Start time (hrs)* enter "2".
11. For the *End time (hrs)* enter "16".
12. For the *Timestep (s)* enter "5".
13. From the list on the left, select "Water Level".
14. For the *Initial water level (m)* enter "265.5".
15. From the list on the left, select "BC".
16. From the *BC event name* drop-down, select "100 year".
17. Click the **OK** button to close the *TUFLOW 2D Model Control* dialog.

4 Saving a Project File

Before running TUFLOW, the project should be saved under a new name:

1. Select the *File* | **Save As…** menu command to bring up the *Save As* dialog.
2. From the *Save as type* drop-down, select "Project Files (*.sms)".
3. For the *File name*, enter "Cimarron2d.sms".
4. Click the **Save** button to save the new project file and exit the *Save As* dialog.

5 Running TUFLOW

TUFLOW can be launched from inside of SMS. Make sure that your TUFLOW license is active and available to 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 " 100year_20m" simulation and select the **Export TUFLOW files** context menu command.

This creates a directory named "TUFLOW" where the files will be written. The directory structure models are described in the *TUFLOW User's Manual*.

2. Right-click on the " 100year_20m" simulation and select the **Launch TUFLOW** context menu command to bring up an external console window in which TUFLOW will run. Depending on the speed of the computer being used, this process can take several minutes to complete.
3. Click the **OK** button when the dialog indicates the simulation is finished.

The dialog may appear behind the SMS window and the TUFLOW window, depending on what other windows have been accessed while waiting for the simulation to finish.

6 Using Log and Check Files

TUFLOW generates several files that can be useful for locating problems in a model. In the *data_file\TUFLOW\runs\log* directory, there should be a file named "100year_20m.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 the *File* | **View Data File…** menu command to bring up the *Open* dialog.
2. Browse to the *data files\TUFLOW\runs\log* and select the “100year_20m.tlf” image file.
3. Click the **Open** button to bring up the *View Data File* dialog. If the *Never ask this again* option had previously been turned on, this dialog will not appear. In that case, skip to step 5.
4. Select the desired text editor from the *Open with* drop-down and click the **OK** button to close the *View Data File* dialog.
5. Scroll to the bottom of the file.

The bottom of this file will report if the run finished, whether the simulation was stable, and 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 “100year_20m_messages.mif” and “100year_20m_messages.mid”.

6. When done, click the **Close** button in the text editor and return to SMS.

To view these files in SMS:

7. Click the **Open** macro to bring up the *Open* dialog.
8. Select “100year_20m_messages.mif” and click **Open** to bring up the *Mif/Mid import* dialog.

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

9. In the *Read As* section, from the drop-down, select “GIS layer”.
10. Click the **OK button** to close the *Mif/Mid import* dialog and import the MIF file. If necessary, specify the projection.

The file will display various points with warnings. A number of the warnings refer to points being outside the grid domain; this is fine and can be ignored.

11. In the Project Explorer, uncheck the “100yr_20m_messages.mif” shapefile to hide the layer.

For more information on using the GIS module, see the “GIS” tutorial.

7 Viewing the Solution

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

The results folder contains several different types of SMS files which contain a 2D mesh and accompanying solutions, representing the 2D portions of the model.

To view the solution files from within SMS:

1. Click the **Open** macro to bring up the *Open* dialog.
2. Browse to the *data_files\TUFLOW\results* folder and select "100year_20m.xmdf.sup".
3. Click the **Open** button to import the file and exit the *Open* dialog.

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

4. If a dialog asks to define a projection, select the default and click the **OK** button.
5. If a dialog asks to replace existing material definitions, click the **No** button.
6. If a dialog asks for time units, select "hours".
7. In the Project Explorer, turn off the " Map Data" and " Scatter Data" folders.
8. Turn on and select the " Mesh Data" folder to make it active.

The mesh will be contoured according to the selected dataset and time step. The final time-step should appear similar to Figure 7. If it doesn't, some display options may need to be adjusted to more closely resemble the figure, such as reversing the color ramp and turning on vectors.

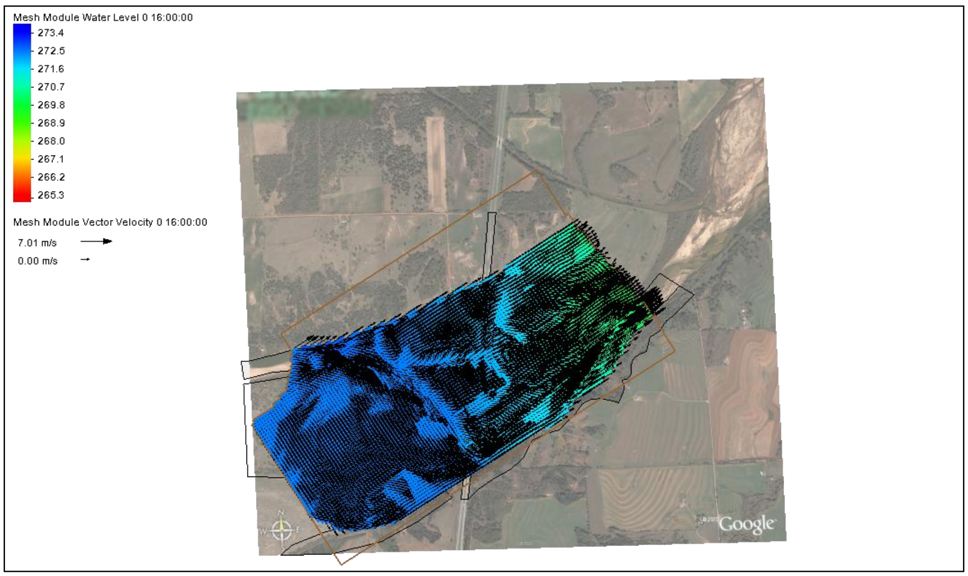


Figure 7 Final time step

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 Conclusion

This concludes the "TUFLOW 2D Hydrodynamics" tutorial.

Feel free to continue experimenting with the effects of changing material properties. Create new material sets (perhaps 20% rougher, for example) and new simulations to contain them. This prevents TUFLOW from overwriting previous solutions, allowing comparison of the results. When done experimenting, exit the program.