SMS 13.0 Tutorial

TUFLOW 2D Hydrodynamics

O...

T...
tw...
co...
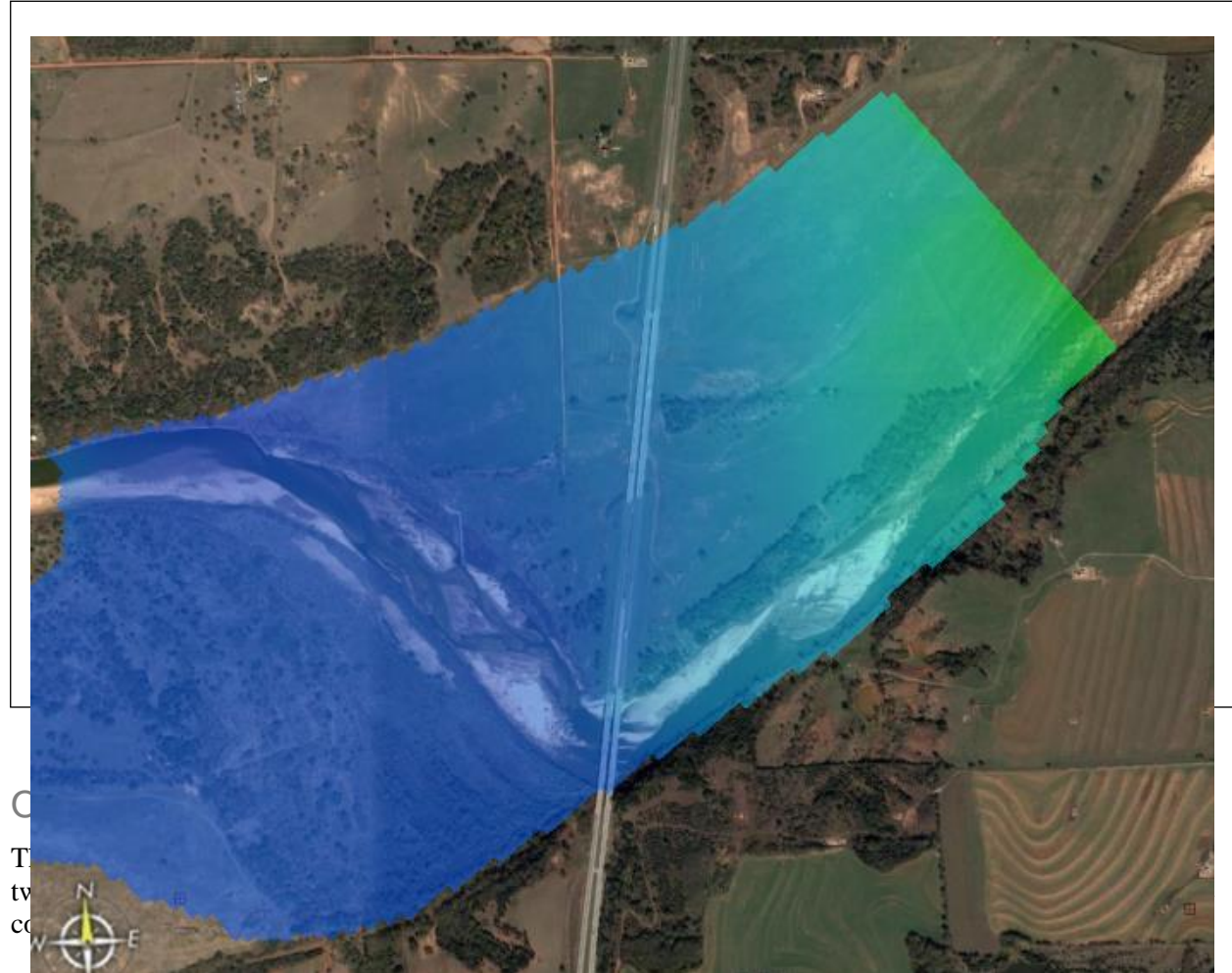
Prerequisites

- Overview Tutorial

Requirements

- TUFLOW 2D
- Map Module
- Scatter Module
- Cartesian Grid Module
- Mesh Module

Time

- 20-30 minutes

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 *File |* **Delete All**.

3. Click **Yes** to clear all data in SMS.

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 the grid, see the "Creating a Cell-Centered Grid" tutorial.

1. Select *File* / **Open…** to bring up the *Open* dialog.

2. Browse to the *data files* folder for this tutorial and select "start.sms".

3. Click **Open** to import the project file and exit the *Open* dialog.

---

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

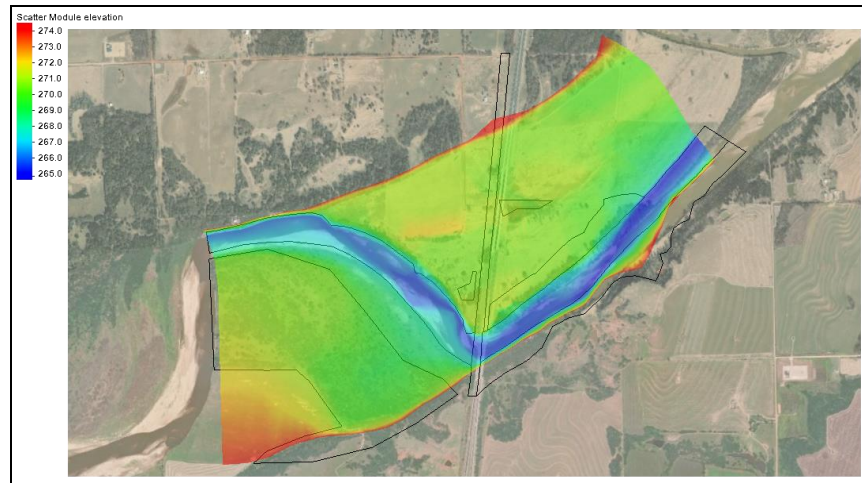The project should appear similar to Figure 1.

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

An area property coverage defines the material zones of the grid. This can be done 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 inputs 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 under the Map Data.

To create the material zone polygons in the map module:

1. Right-click on "🗂 Map Data" and select **New Coverage** to bring up the *New Coverage* dialog.

2. In the *Coverage Type* section, select *Generic* | **Area Property**.

3. Enter "materials" as the *Coverage Name*.

4. Click **OK** to exit the *New Coverage* dialog and create the new coverage.

5. Select "🔿 materials" to make it active.

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

6. Select " 🖼 materials.shp" under " 🌐 GIS Data".

7. Select *Mapping* | **Shapes → Feature Objects** to bring up the *Step 1 of 2* page of
    the *GIS to Feature Objects Wizard* dialog.

8. Select "materials" from the *Select a coverage for mapping* drop-down.

9. Click **Next** 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 **Finish** to close the *GIS to Feature Objects Wizard.'*

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 to a
default material value. The default material for the simulation is grassland. This material
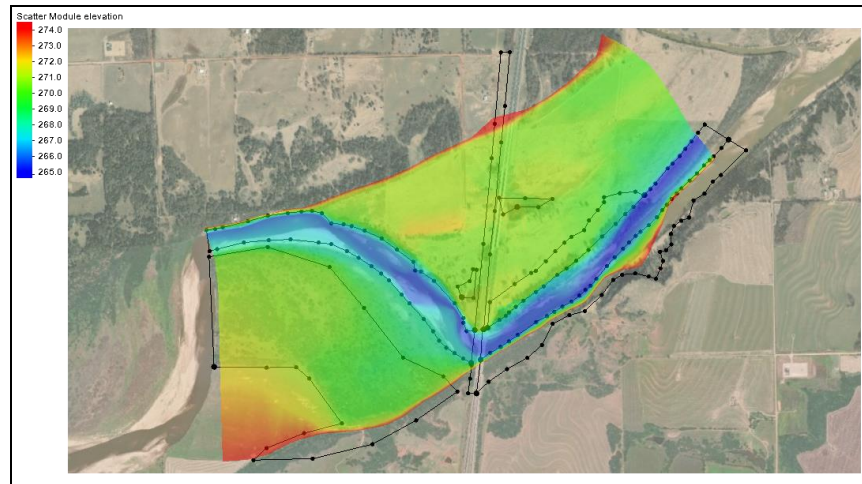hasn't yet been created since it was not part of the area property coverage.

Figure 2   The shapefile polygons do not cover the entire domain

To create this material:

12. Select *Edit* / **Materials Data** to bring up the *Materials Data* dialog.

13. At the bottom of the *Materials* section, click **New** to create a new "material 06".

14. Double-click on "material 06" and rename the material to "grasslands".

15. Click **OK** 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 "⬤ BC" and select it to make it active.

2. Right-click on "⬤ BC" and select *Type | Models | TUFLOW* | **1D-2D BCs and Links**.

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.
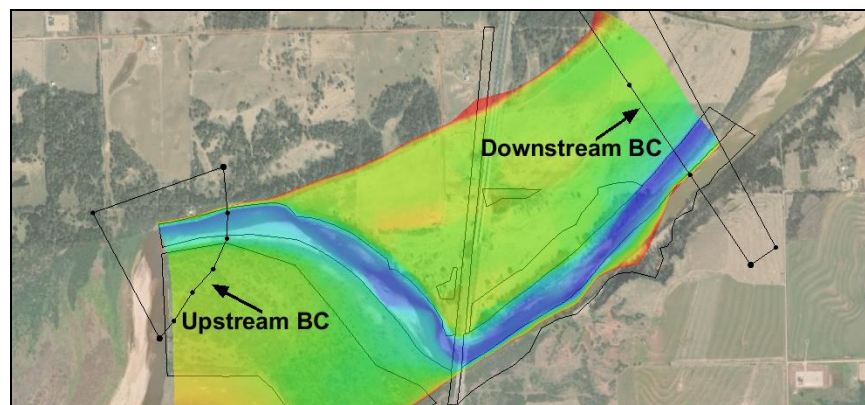

Figure 3   Locations of upstream and downstream BCs

1. Using the **Select Feature Arc** tool, double-click on the upstream BC arc to bring up the *Boundary Conditions* dialog.

2. Select "Flow vs Time (QT)" from the *Type* drop-down.

3. In the *Events* section, click **Edit Events...** to open the *TUFLOW BC Events* dialog.

4. In the *Events* section, click **Add** to create an event titled "new_event".

5. Double-click on "new_event" and enter "100 year" as the new name.

6. Click **OK** to close the *TUFLOW BC Events* dialog.

7. Select "100 year" from the list of events and click **Curve undefined** to bring up the *XY Series Editor* dialog.

8. Click **Import…** to bring up the *Open File* dialog.

9. Select "upstream.xys" and click **Open** 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 **OK** to close the *XY Series Editor* dialog.

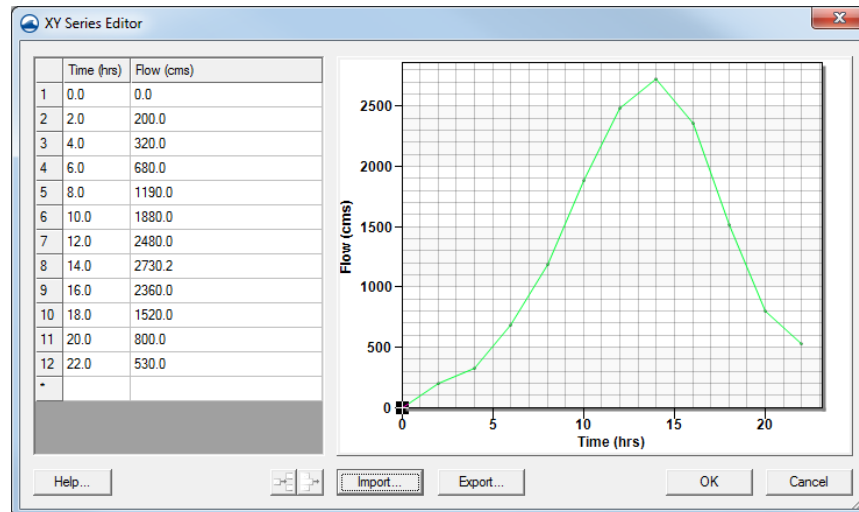11. Click **OK** to close the *Boundary Conditions* dialog.

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** tool, double-click the downstream boundary condition 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 "100 year" from the list of events and click **Curve undefined** to bring up the *XY Series Editor* dialog.

4. Click **Import…** to bring up the *Open File* dialog.

5. Select "downstream.xys" and click **Open** to import the data and exit the *Open File* dialog.

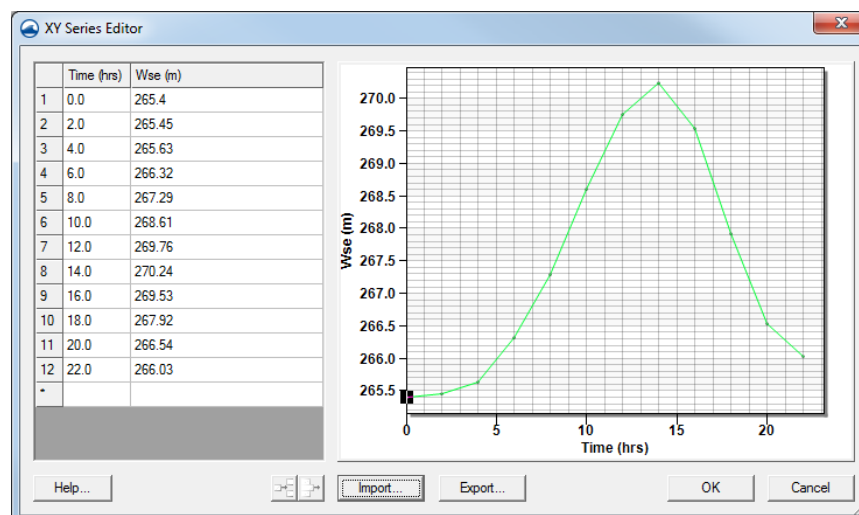The dialog should appear similar to Figure 5.

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).

6. Click **OK** to close the *XY Series Editor* dialog.

7. Click **OK** to close the *Boundary Conditions* dialog.

2.5   Grid Activity Zones

In this simulation, the grid will use 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** 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 select "Inactive – not in mesh" from the drop-down.

3. Click **OK** to close the *Boundary Conditions* dialog.

4. Repeat steps 1–3 for the upstream polygon.

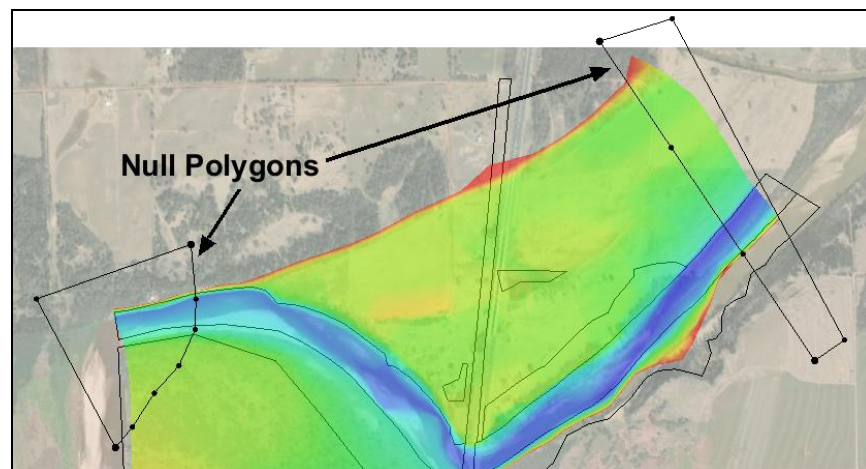5. Click anywhere outside of the polygons to deselect all polygons.

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 that each include 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. Right-click in an empty space at the bottom of the Project Explorer and select *New Simulation* / **TUFLOW**.

This creates 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 "🔵 Sim" and select **Rename**.

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 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 setup 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 "20m_geo" and press *Enter* to set the new name.

4. Drag "🟢 materials", "🟢 BC", and "▦ 20m" under "🧩 20m_geo".

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 **Grid Options…** to bring up the *Grid Options* dialog.

6. In the *Materials* section, select "Specify using area property coverage(s)".

7. Select "grasslands" from the *Default material* drop-down.

8. In the *Cell codes* section, select "Specify using BC coverage(s)".

9. Select "water cell" from the *Default code* drop-down.

10. Click **OK** 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 " Material Sets" and select **New Material Set** to create a new " Material Set".

2. Right-click on " Material Set" and select **Properties** to bring up the *TUFLOW Material Properties* dialog.

3. Select "channel" from the list on the left.

4. Enter "0.03" as the *n* on the right.

This is the Manning's *n* value.

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

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

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

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. In the "TUFLOW 1D/2D" tutorial, a model is 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 " 20m_geo" onto the " 100year_20m" simulation in the Project Explorer.

2. Right-click on " 100year_20m" and select **2D Model Control…** to open the *TUFLOW 2D Model Control* dialog.

3. Select "Output Control" from the list on the left.

4. In the *Map Output* section, select "SMS 2dm" from the *Format type* drop-down.

5. Enter "0" as the Start *Time*.

6. Enter "900" as the *Interval*.

7. In the *Output Datasets* section, turn on *Depth*, *Water Level*, *Flow Vectors*, and *Velocity Vectors*.

8. 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.

9. Select "Time" from the list on the left.

10. Enter "2" as the *Start Time (hrs)*.

11. Enter "16" as the *End Time (hrs)*.

12. Enter "5" as the *Time step (s)*.

13. Select "Water Level" from the list on the left.

14. Enter "265.5" as the *Initial water level (m)*.

15. Select "BC" from the list on the left.

16. Select "100 year" from the *BC event name* drop-down.

17. Click **OK** 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 *File* / **Save As…** to bring up the *Save* dialog.

2. Select "Project Files (*.sms)" from the *Save as type* drop-down.

3. Enter "Cimarron2d.sms" as the *File name* and click **Save** to save the new project file and exit the *Save* dialog.

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 "⊙ 100year_20m" and select **Export TUFLOW files**.

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 "⊙ 100year_20m" and select **Launch TUFLOW** 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 **OK** 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 *File* / **View Data File…** to bring up the *Open* dialog.

2. Browse to the *data files\TUFLOW\runs\log* folder and select "100year_20m.tlf".

3. Click **Open** 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 **OK** 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, **Close** the text editor and return to SMS.

To view these files in SMS:

7. Click **Open** 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, select "GIS layer" from the drop-down.

10. Click **OK** to close the *Mif/Mid import* dialog and import the MIF file.

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. Uncheck the "100yr_20m_messages.mif" in the project explorer 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 which 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 **Open** 📂 to bring up the *Open* dialog.

2. Browse to the *data files\TUFLOW\results* folder and select "100year_20m.xmdf.sup".

3. Click **Open** 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 replace existing material definitions, click **No**.

5. If a dialog asks for time units, select "hours".

6. Turn off " Map Data"and " Scatter Data" in the Project Explorer.

7. Turn on and select " Mesh Data" 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.
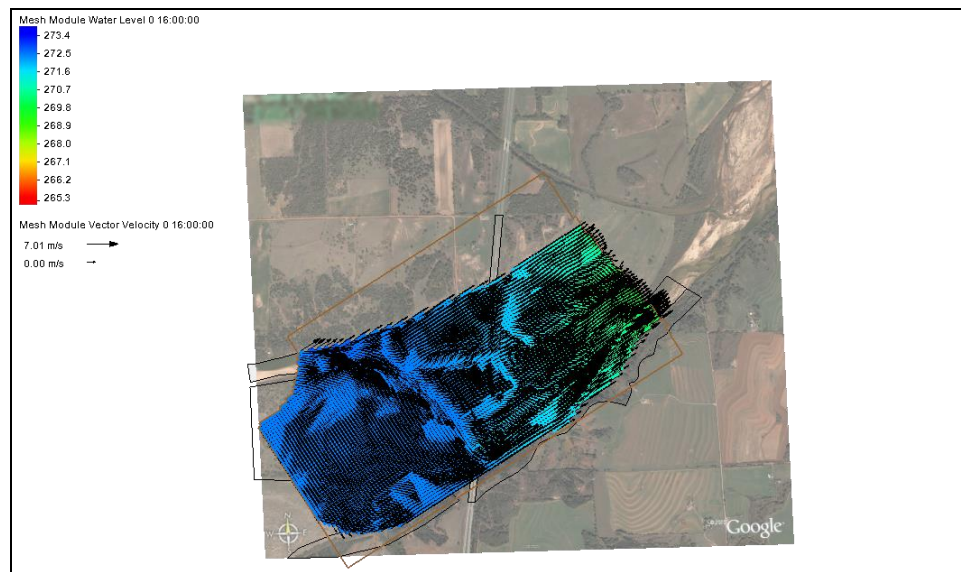

Figure 7   Final time step

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.