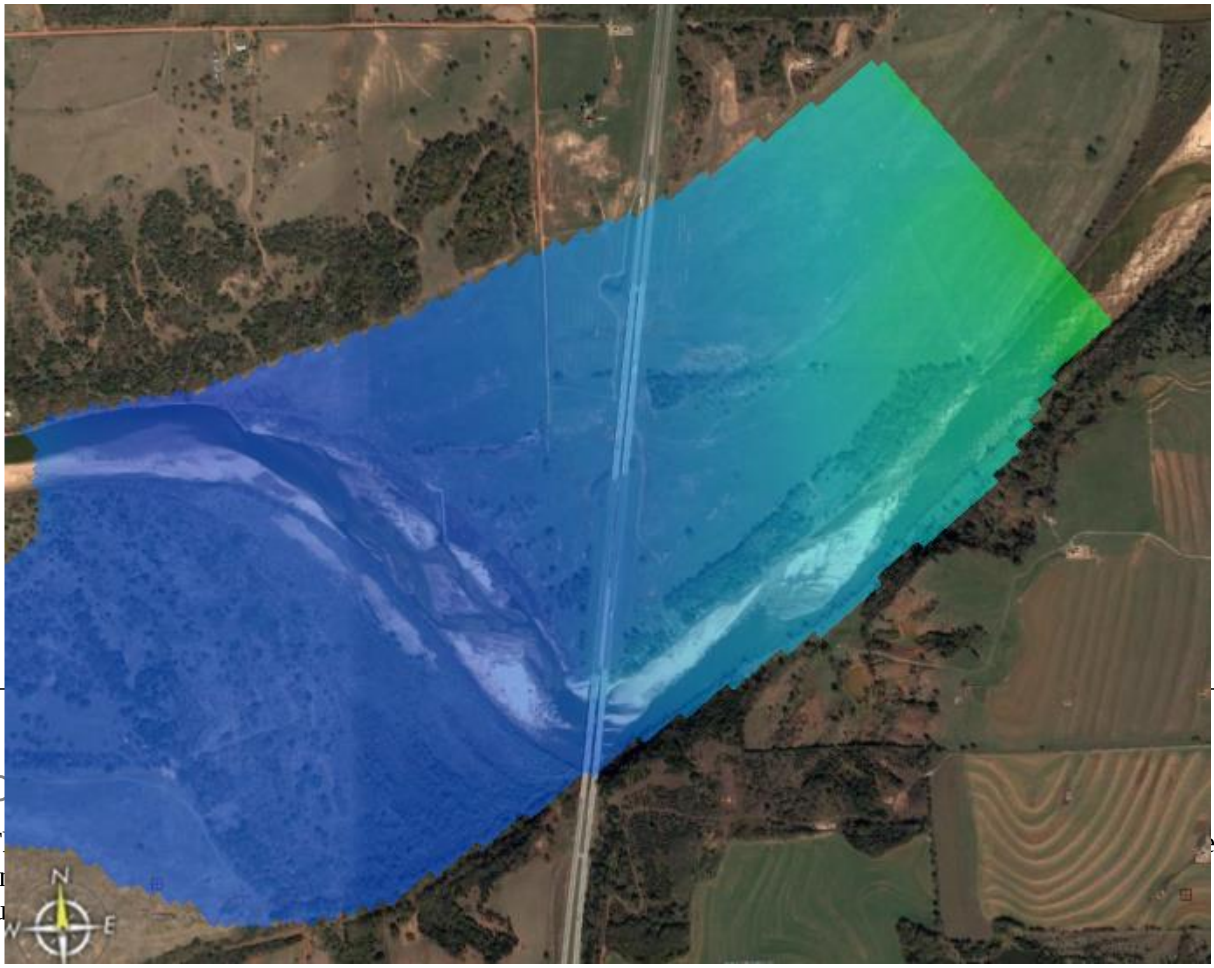
SMS

v. 12.3

SMS 12.3 Tutorial

TUFLOW-2D Hydrodynamics



Prerequisites

- Overview Tutorial

Requirements

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

Time

- 60-90 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.

2 Background Data

SMS modeling studies requires or uses several types of data. This data includes:

- Geographic (location) and topographic (elevation) data. Note that all units in TUFLOW must be metric.
- Maps and images
- Land use data (may be extracted from images)
- Boundary conditions.

Start by loading the first two items of data.

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

2.1 Bathymetry and Background Data

Topographic data in SMS is managed by the scatter module either as scattered datasets or triangulated irregular networks (TIN). SMS uses this data as the source for elevation data in the study area.

To open the scatter set data:

1. Select *File* | **Open…** to bring up the *Open* dialog.
2. Browse to the *data files* folder for this tutorial and select "Cimarron Survey 2005.h5".
3. Click **Open** to import the H5 file and exit the **Open** dialog.

The screen will refresh, showing a set of scattered data points (Figure 1).

Figure 1 Imported survey scatter set

An image of the study location is often useful when building a numeric model. An image for the study site was generated using Google Earth Pro.

To import this file, do the following:

4. Select *File* | **Open…** to bring up the *Open* dialog.
5. Select file "ge_highres.jpg" and click **Open** to import the image and exit the *Open* dialog.
6. **Frame** the project.

The project should appear similar to Figure 2.

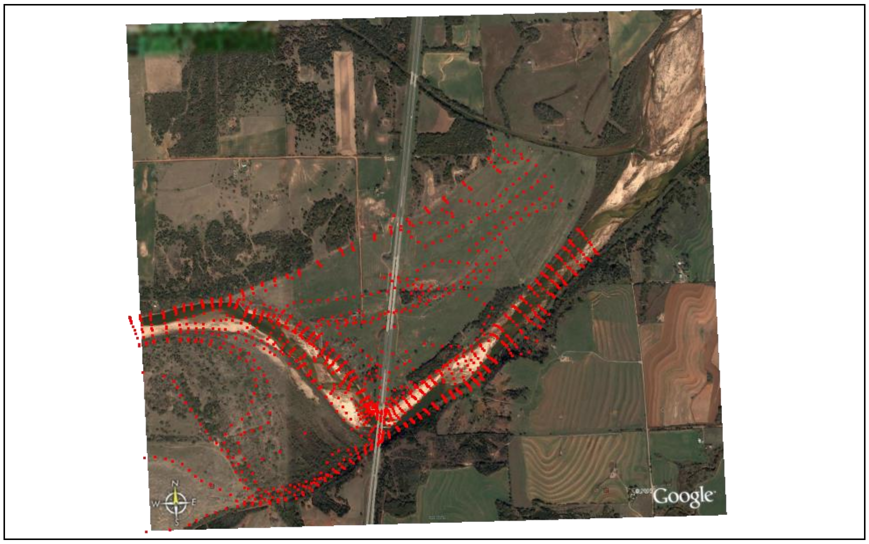


Figure 2 Scatter set with background image

2.2 Modifying the Display

Now that the initial data is loaded, adjust the display. Items loaded into SMS can be turned on and off by clicking in the box to the left of the item in the Project Explorer. During this tutorial, images can be turned on or off to reference the location of features or to simplify the display.

Make sure the following display settings are being used.

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* and turn on *Boundary* and *Contours*.
4. On the *Contours* tab, in the *Contour Method* section, select “Color Fill” from the first drop-down.
5. Click **Color Ramp…** to bring up the *Color Options* dialog.
6. In the *Palette Preview* section, make sure the dark blue is on the *Min* side of the palette. If it is not, click **Reverse**.
7. Click **OK** to close the *Color Options* dialog.
8. Enter “50” as the *Transparency*.
9. Click **OK** to close the *Display Options* dialog.

The project should appear similar to Figure 3.

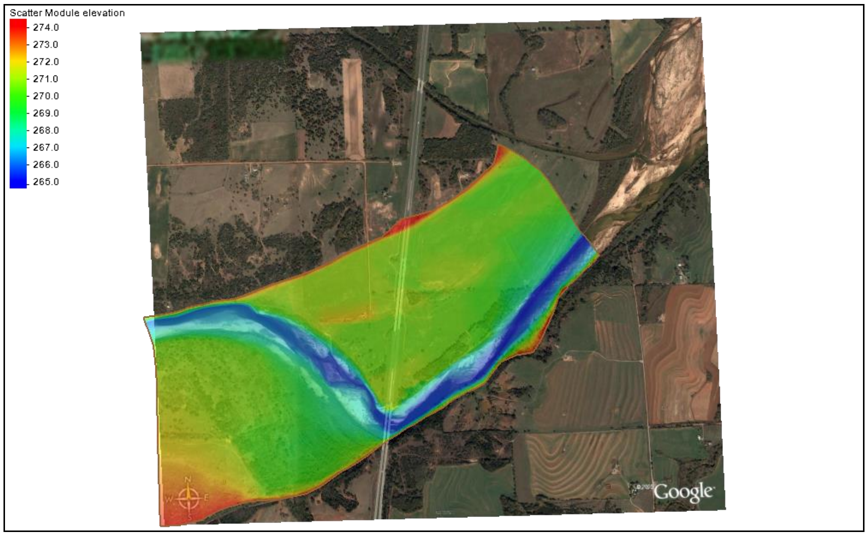


Figure 3 Contours with river portion showing blue

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

To create the grid in this example:

1. Right-click on " Area Property" in the Project Explorer and select **Rename**.
2. Enter "TUFLOW grid" and press *Enter* to set the new name.
3. Right-click on " TUFLOW grid" and select *Type* | *Models* | *TUFLOW* | **2D Grid Extents**.
4. Using the **Create 2-D Grid Frame** tool, create a grid frame as indicated in Figure 4.

The size and positioning of the frame will be adjusted later, so it does not need to be exact. To edit the location/size of the grid frame after creating it:

5. Using the **Select 2-D Grid Frame** tool, select the box in the center of the grid frame to expose the editing handles.
6. Drag the handles on each side and corner of the grid frame to adjust the size of the grid frame to closer approximate the grid frame in Figure 4.

The circle near one of the grid frame corners can be used to rotate the grid frame, if needed.

7. Select *Feature Objects* | **Map** → **2D Grid** to bring up the *Map → 2D Grid* dialog.

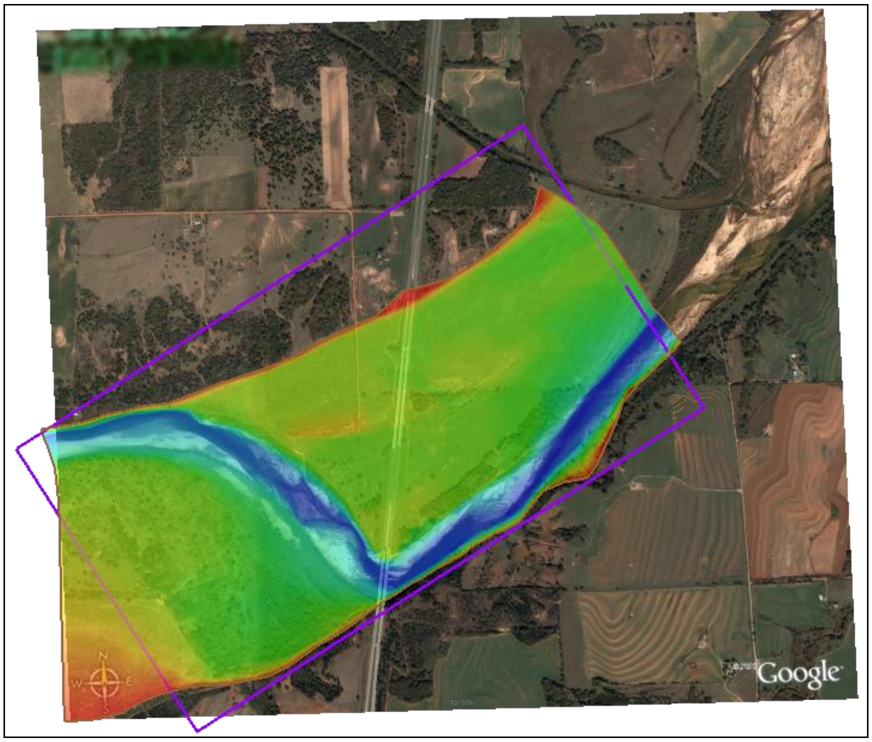


Figure 4 Creation of the grid frame

The first grid created will be very coarse. Starting with a coarse grid is useful to get quick model results and find problems quickly. If necessary, it is easy to create a finer grid after some initial runs.

8. In the *I Cell Options* section, enter "20.0" as the C*ell Size*.

This automatically changes the *J Cell Options* to match. The *J Cell Options* cannot be directly edited here.

9. In the *Elevation options* section, select "Scatter Set" from the *Source* drop-down.
10. Click **Select…** to bring up the *Interpolation* dialog.
11. In the *Scatter Set To Interpolate From* section, make sure that "elevation" is selected.
12. In the *Interpolation Options* section, select "Single Value" from the *Extrapolation* drop-down.
13. Enter "278.0" as the *Single Value*.

SMS assigns all cells not inside the TIN to this value. 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 **OK** to close the *Interpolation* dialog.
15. Click **OK** to close the *Map → 2D Grid* dialog.

This creates a new " TUFLOW grid Grid" under " Cartesian Grid Data" in the Project Explorer.

16. Right-click on " TUFLOW grid Grid" and select **Rename**.
17. Enter "20m" and press *Enter* to set the new name.

The project should appear similar to Figure 5.

Figure 5 The newly-created grid

3.2 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 an ESRI 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.

To import the area properties for this example and get the data into 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. Click **Open** to bring up the *Open* dialog.
7. Select "materials.shp" and click **Open** to exit the Open dialog and import the shapefile into the GIS module.
8. Select " materials.shp" under " GIS Data" to make it active.
9. Select *Mapping* | **Shapes → Feature Objects** to bring up the *GIS to Feature Objects Wizard* dialog.
10. Select "materials" from the *Select a coverage for mapping* drop-down.

11. Click **Next** to go to the *Step 1 of 1* page of the *GIS to Feature Objects Wizard* dialog.
12. In the *MATNAME* column on the *Mapping* row, select “Material” from the drop-down.
13. Click **Finish** to close the *GIS to Feature Objects Wizard*.

Notice that the area property coverage contains polygons (they may be partially obscured below the grid), 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 been created since it was not part of the area property coverage.

To create this material:

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

3.3 2D Boundary Condition Coverage

The boundary conditions (BC) for the model need to be specified. This model will include 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.

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 for a project.

A tidal curve and a storm surge curve can both be specified at one location and TUFLOW will sum them to generate a combined water surface elevation curve. In this case, the tidal curve and the storm surge are separate components, each comprised of parameters which generally include a time series curve.

Individual 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 create the upstream boundary condition arc and assign boundary conditions:

1. Right-click on “ Map Data” and select **New Coverage** to bring up the *New Coverage* dialog.
2. In the *Coverage Type* section, select *Models* | *TUFLOW* | **1D–2D BCs and Links**.
3. Enter “ BC” as the *Coverage Name*.
4. Click **OK** to close the *New Coverage* dialog.
5. Select “BC” to make it active.
6. Using the **Create Feature Arc** tool, click out an arc at the location labeled “Upstream BC” in Figure 6.

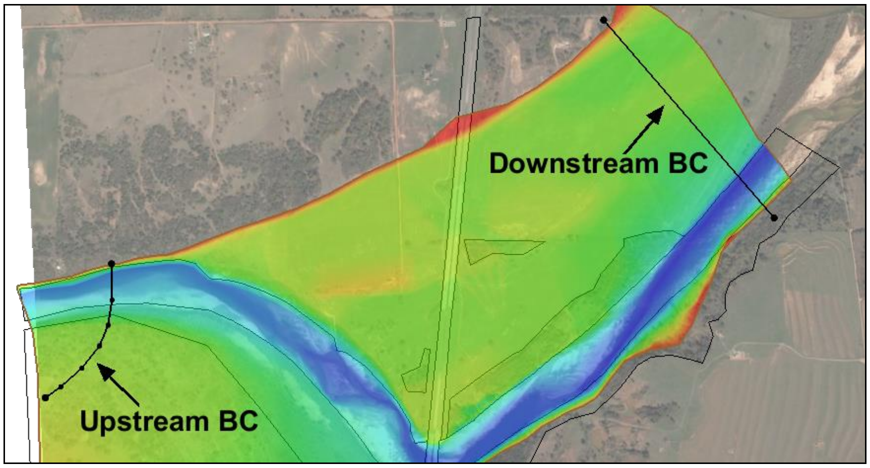


Figure 6 Locations of upstream and downstream BCs

Inflow boundary arcs should be created such that constant water surface (head) can be assumed along the arc. The arc as shown in Figure 6 is angled upstream in the floodplain as a better approximation of the correct equal head condition.

7. Using the **Select Feature Arc** tool, double-click on the upstream BC arc to bring up the Boundary Conditions dialog.
8. Select "Flow vs Time (QT)" from the *Type* drop-down.
9. In the *Events* section, click **Edit Events...** to open the *TUFLOW BC Events* dialog.
10. In the *Events* section, click **Add** to create an event titled "new_event".
11. Double-click on "new_event" and enter "100 year" as the new name.
12. Click **OK** to close the *TUFLOW BC Events* dialog.
13. Select "100 year" from the list of events and click **Curve undefined** to bring up the *XY Series Editor* dialog.

To copy the values needed for this tutorial:

14. Outside of SMS, open the file "bc.xls" from the tutorial *data files* folder in a spreadsheet program
15. In the spreadsheet program, select cells *A2* through *B13* and copy them (*Ctrl-C*).
16. In SMS, click in the blank cell in the *Time (hrs)* column and paste (*Ctrl-V*) copied data into the spreadsheet.

The values from the spreadsheet should now be listed in both columns, and the graph on the right will show a steep curve (Figure 7).

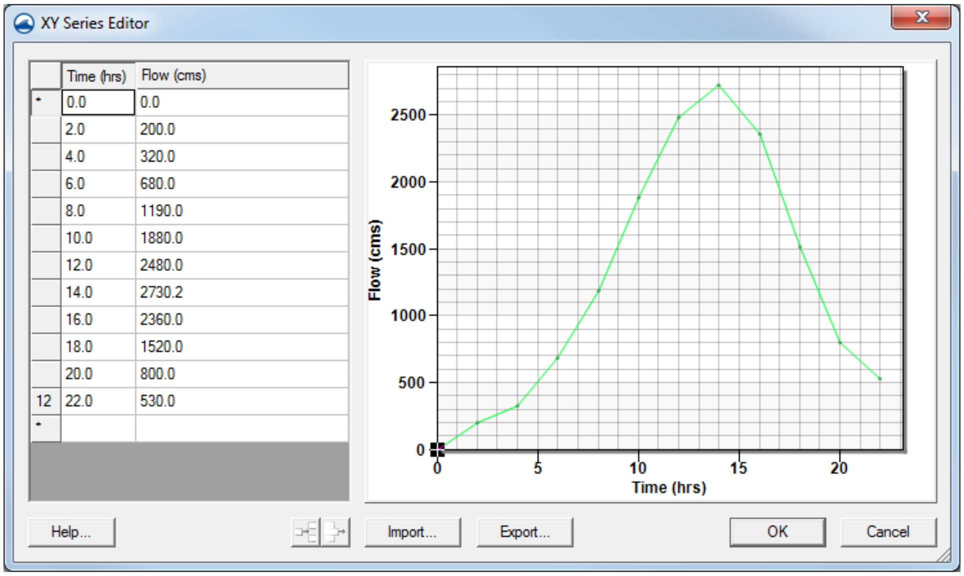


Figure 7 XY Series Editor dialog showing curve

17. Click **OK** to close the *XY Series Editor* dialog.
18. Click **OK** to close the *Boundary Conditions* dialog.

To create the downstream boundary arc and setup the boundary condition:

1. Using the **Create Feature Arc** tool, click out an arc across the downstream portion of the model as shown in Figure 6.

Note that the arc can have as many or few vertices as desired. Since it is unknown how much of the model will be wet, create an arc across the whole model and TUFLOW will only use the wet portions of the boundary.

2. Using the **Select Feature Arc** tool, double-click the downstream boundary condition arc to bring up the *Boundary Conditions* dialog.
3. In the *Events* section, select “Wse vs Time (HT)” from the *Type* drop-down.
4. Select “100 year” from the list of events and click **Curve undefined** to bring up the *XY Series Editor* dialog.
5. Outside SMS, in the “bc.xls” file in the spreadsheet program, select cells *A2* through *A13* and copy them (*Ctrl-C*).
6. In SMS, select the empty cell at the top of the *Time (hrs)* column and paste (*Ctrl-V*) the values.
7. In the spreadsheet program, select cells *C2* through *C13* and copy them.
8. In SMS, select the empty cell at the top of the *Wse (m)* column and paste (*Ctrl-V*) the values.

The values from the spreadsheet should now be listed in both columns, and the graph on the right will show a steep curve (Figure 8).

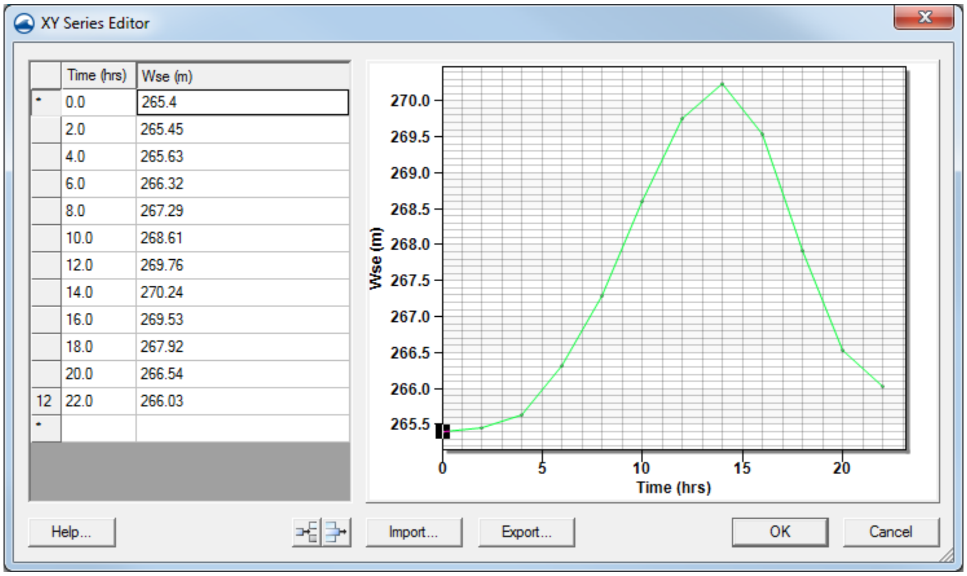


Figure 8 XY Series editor for downstream BC

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

Earlier in the tutorial, it was specified that 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.

This can be specified 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 polygons.

To create the inactive polygons:

1. Using the **Create Feature Arc** tool, create an arc starting at one end of the downstream boundary condition arc that encloses all of the grid downstream of the arc and closes on the other end of the downstream boundary condition arc (Figure 9).
2. Repeat this process to define a polygon on the upstream side of the upstream boundary condition as shown in Figure 9.

Enclosed arcs are not automatically converted into polygons in SMS, so they must be built by doing the following:

3. Select *Feature Objects* | **Build Polygons**.

Next, assign boundary conditions to the polygons by doing the following:

4. Using the **Select Feature Polygon** tool, double-click on the downstream polygon to bring up the *Boundary Conditions* dialog.
5. In the *Options* section, turn on *Set cell code* and select “Inactive – not in mesh” from the drop-down.
6. Click **OK** to close the *Boundary Conditions* dialog.
7. Repeat steps 4–6 for the upstream polygon.

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

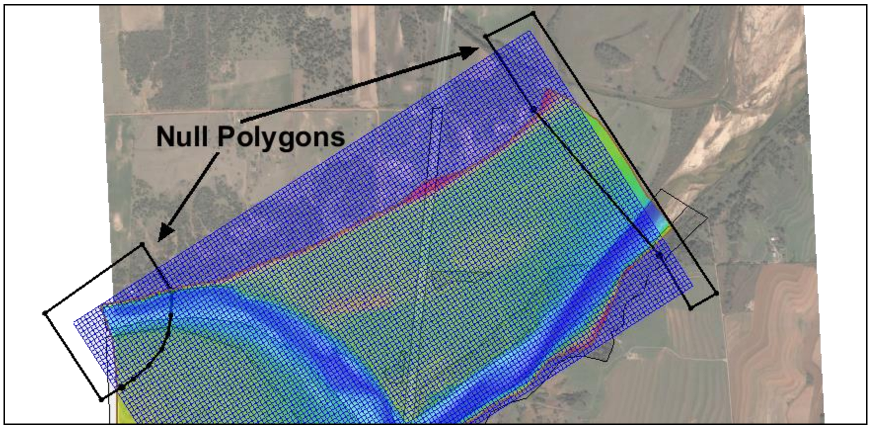


Figure 9 Null polygons

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 in this tutorial to use in TUFLOW simulations. 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 the empty part 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 tree 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.

4.1 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" and "BC" under "20m_geo".
5. Drag "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:

6. Right-click on "20m_geo" and select **Grid Options…** to bring up the *Grid Options* dialog.
7. In the *Materials* section, select "Specify using area property coverage(s)".
8. Select "grasslands" from the *Default material* drop-down.
9. In the *Cell codes* section, select "Specify using BC coverage(s)".
10. Select "water cell" from the *Default code* drop-down.
11. Click **OK** to exit the *Grid Options* dialog.

4.2 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

4.3 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 Leve” from the list on the left.
14. Enter “265.5” as the *Initial water level (m)*.
15. Turn on *Override default instability level (10m above highest elevation)* and enter “285.0” as the *Instability level (m)*.
16. Select “BC” from the list on the left.
17. Select “100 year” from the *BC event name* drop-down.
18. Click **OK** to close the *TUFLOW 2D Model Control* dialog.

5 Saving a Project File

To save all the data as a project file for use in a later session:

1. Select *File* | **Save New Project…** to bring up the *Save* dialog.
2. Select “Project Files (*.sms)” from the *Save as type* drop-down.

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

6 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 may have been accessed while waiting for the simulation to finish.

7 Using Log and Check Files

TUFLOW generates several files that can be useful for locating problems in a model. In the *TUFLOW directory under \runs\log*, 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* directory 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”.

To view these files in SMS:

1. Click **Open** to bring up the *Open* dialog.
2. 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.

3. In the *Read As* section, select “GIS layer” from the drop-down.
4. Click **OK** to close the *Mif/Mid import* dialog and import the MIF file.

Because there are no errors in these files, nothing will happen. If the simulation had any errors or warnings, they would show up in this file. Otherwise, the file is empty (as in this case).

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

8 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 files which have “_1d” after the simulation name are results for the 1D portions of the model. They are not used in this tutorial.

The results folder contains a *.2dm, *.mat, *.sup, and several *.dat files. These are 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”, “ Scatter Data”, and “ Cartesian Grid Data” in the Project Explorer.
7. Turn on and select “ Mesh Data” to make it active.
8. Click **Display Options** to bring up the *Display Option* dialog.
9. Select “2D Mesh” from the list on the left.
10. On the *2D Mesh* tab, turn on *Contours* and *Vectors* and turn off *Elements* and *Nodes*.
11. On the *Contours* tab, in the *Contour method* section, select “Color Fill” from the first drop-down.
12. Click **OK** to close the *Display Options* dialog.

The mesh will be contoured according to the selected dataset and time step. The final time step should appear similar to Figure 10.

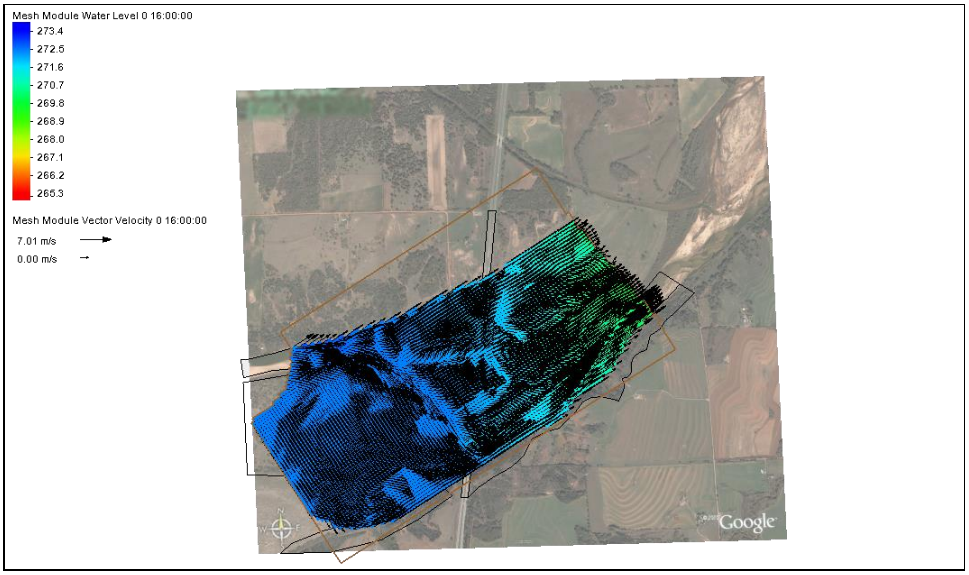


Figure 10 Final time step

At this point, any of the techniques demonstrated in the post-processing tutorial can be used to visualize the TUFLOW results, including film loops and observation plots.

9 Including the Roadway in the Model

The bathymetry data did not adequately represent the road embankment. Even if the road was represented in the TIN, it is unlikely the coarse grid would have represented it well. Use of the higher elevations can be forced by using a geometry modification coverage. TUFLOW will use the same grid input files but modify the grid based upon these modifications. The bridge and relief openings will not be represented in the geometry modification coverage. These openings will be modeled with the assumption that the water does not reach the bridge decks and go into pressure flow.

A geometry modification coverage can contain arcs or polygons and is used to override previously defined grid elevations. For an arc, the elevations at the nodes of the arc (at the ends) are interpolated along the arc while the elevations at vertices are ignored. Vertices are only used to define the shape of the arc. To specify varying elevations along a path, split the arc into multiple pieces. A polygon can be used to raise/lower whole regions of cells. The elevation used for a polygon can be set by double-clicking on the arc.

To define the roadway arc, first define a new coverage:

1. Right-click on " Map Data" and select **New Coverage** to bring up the *New Coverage* dialog.
2. In the *Coverage Type* section, select *Models* | *TUFLOW* | **1D–2D BCs and Links**.
3. Enter "Roadway" as the *Coverage Name.*
4. Click **OK** to close the *New Coverage* dialog.

5. Select "Roadway" to make it active

Now add feature objects to the coverage:

6. Turn off "Mesh Data" in the Project Explorer so the roadway is visible.
7. Using the **Create Feature Arc** tool, click out two arcs for the road embankments as shown in Figure 11.
8. Using the **Select Feature Point** tool, select the top node and enter "274.5" in the *Z* field at the top of the SMS window.
9. Repeat step 8 for each of the other three nodes, entering "274.0", "273.5", and "273.0", as indicated to Figure 11.

Figure 11 Roadway embankment arc and elevations

10 New Geometry Component and Simulation

Rather than change the existing simulation, create a new simulation that includes the roadway. This is a powerful tool which allows multiple configurations to share some of the input files and prevents overwriting earlier solutions. Since the roadway coverage needs to be added to a geometry component, a new geometry component needs to be created.

To create this component:

1. Right-click on "20m_geo" and select **Duplicate**.
2. Right-click on the new "20m_geo (2)" and select **Rename**.
3. Enter "20m_road" and press *Enter* to set the new name.
4. Drag "Roadway" coverage onto "20m_road".

Similarly, a new simulation needs to be created which uses this geometry component by doing the following:

5. Right-click on "100year_20m" and select **Duplicate**.
6. Right-click on the new "100year_20m (2)" and select **Rename**.

7. Enter “100year_20m_road” and press *Enter* to set the new name.
8. Right-click on the grid component link in the “ 100year_20m_road” simulation labeled “20m_geo” and select **Delete**.
9. When asked to confirm, click **Yes**.

This deletes the link to the grid component, not the component itself.

10. Drag the geometry component “ 20m_road” into the “ 100year_20m_road” simulation.

The new simulation will have the same model control parameters used previously.

11 Save the New Project and Run the New Simulation

1. Select *File* | **Save As...** to bring up the *Save As* dialog.
2. Select “Project Files (*.sms)” from the *Save as type* drop-down.
3. Enter “Cimarron2d_road.sms” as the *File Name*.
4. Click **Save** to save the project under the new name and exit the *Save As* dialog.
5. Right-click on “ 100year_20m_road” and select **Export TUFLOW files**.
6. Right-click on “ 100year_20m_road” simulation 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.
7. Click **OK** when the dialog indicates the simulation is finished.
8. Click **Open** to bring up the *Open* dialog.
9. Browse to the *data files\TUFLOW\results* folder and select “100year_20m_road.xmdf.sup”.
10. Click **Open** to exit the *Open* dialog and bring up the *Select Tree Item for Datasets* dialog.

This dialog asks how to organize the solution datasets in the Project Explorer because there are now multiple meshes as a result of the two model runs.

11. Select “100year_20m_road” and click **OK** to close the *Select Tree Item for Datasets* and to place the solutions under that mesh.
12. Review the “ 100year_20m_road” solution datasets and click through the time steps to see the results.

The final time step should appear similar to Figure 12.

Figure 12 Final time step

The simulation message files may contain negative depths warnings which indicate potential instabilities. These can be reduced by increasing the resolution of the grid and decreasing the time step as required. Complete steps for this will not be given, but it should be straight-forward following the steps outlined in Sections 7 and 8. A grid with 10 meter cells gives solutions without negative depth warnings.

12 Conclusion

This concludes the "TUFLOW 2D Hydrodynamics" tutorial. If desired, experiment 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.