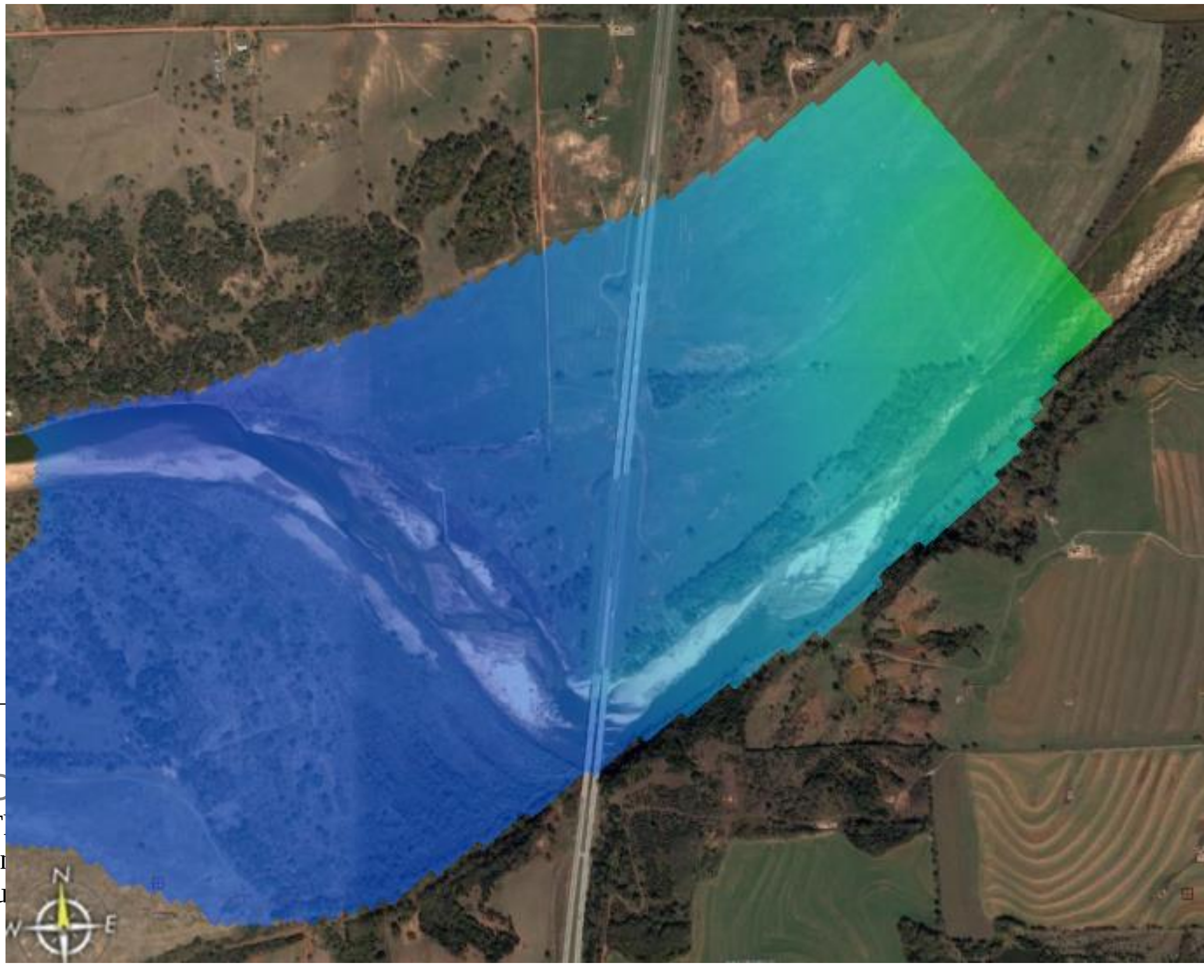
v. 12.2

*SMS 12.2 Tutorial*

# ***TUFLOW-2D Hydrodynamics***




## Prerequisites

- Overview Tutorial

## Requirements

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

## Time

- 60-90 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: www.tuflow.com.

The area used in the tutorial is where the Cimarron River crosses I-35 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:

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

Start by loading the first two items of data.

## 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. Select the file "Cimarron Survey 2005.h5" from the *data files* folder for this tutorial and click **Open**. The screen will refresh, showing a set of scattered data points.

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 open this file:

1. Select *File* | **Open** to bring up the *Open* dialog.
2. Select the file "ge_highres.jpg" and click **Open**.

## 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. Choose *Display* | **Display Options** to bring up the *Display Options* dialog.
2. Select the *Scatter* item from the list on the left then make sure *Points* are turned off and the *Boundary* and *Contours* are turned on.
3. In the *Contours* tab, set the *Contour Method* to "Color Fill" and set the *Transparency* to "50%".
4. If the contour colors in the river portion of the image (the lower contour values) are not blue, click the **Color Ramp** button to open the *Color Options* dialog. Otherwise, skip to step 6.
5. Click the **Reverse** button at the bottom of the *Color Options* dialog.
6. Click **OK** to close the *Display Options* dialog.

# 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 the "Area Property" coverage in the Project Explorer; select **Rename** and change the name to "TUFLOW grid".
2. Right-click on the "TUFLOW grid" coverage and select *Type* | *Models* | *TUFLOW* | **2D Grid Extents**.
3. Switch to the **Map Module** if not already active.
4. Select the **Create 2-D Grid Frame** tool.
5. Create a grid frame around the area shown in Figure 1 by clicking on three of the corners. The size and positioning of the frame will be adjusted later.

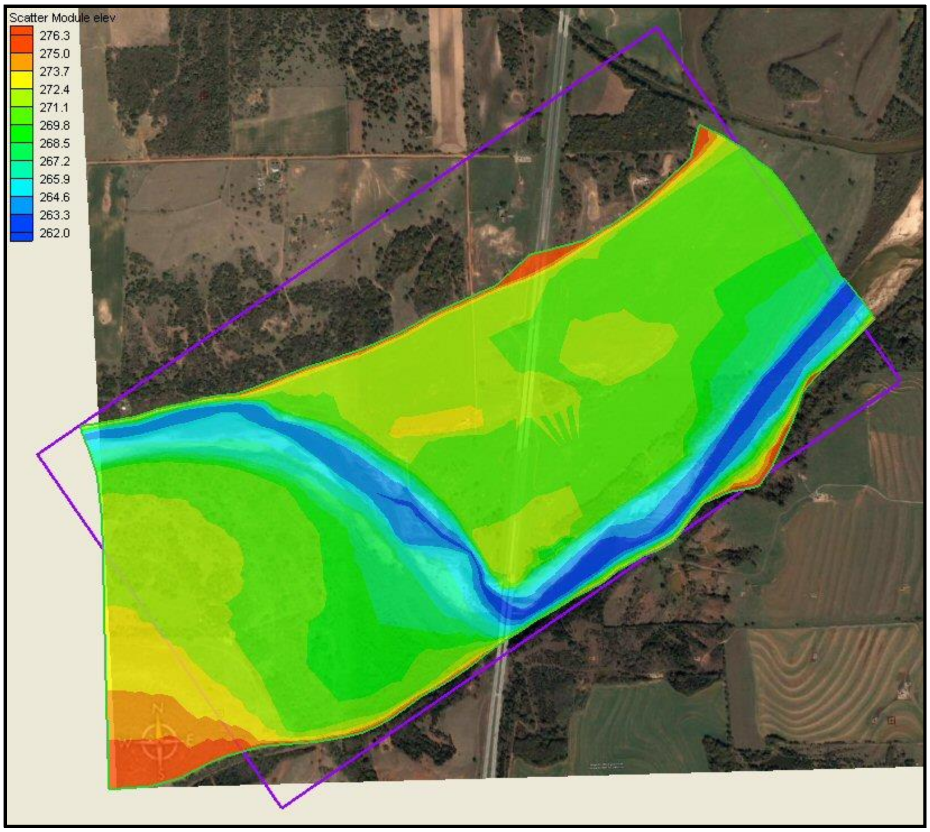



*Figure 1 Creation of the Grid Frame*

6. To edit the location/size of the grid frame after creating it:
   - First select the grid frame by choosing the **Select 2-D Grid Frame** tool and clicking in the box in the center of the grid frame. This exposes the editing handles.
   - 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 1.

- The circle near one of the grid frame corners can be used to rotate the grid frame.

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

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. Set the C*ell Size* to “20” meters in *I Cell Options*. This automatically changes the *J Cell Options* to match. The *J Cell Options* cannot be directly edited here.
9. In the *Elevation Options* section, set the *Source* to “Scatter Set”.
10. Click the **Select** button below the *Source* field to bring up the *Interpolation* dialog.
11. In the *Scatter Set To Interpolate From* section, make sure that “elevation” is selected.
12. Under *Extrapolation*, select “Single Value” from the drop-down menu, and set *Single Value* to “278.0”. 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.
13. Click **OK** to close the *Interpolation* dialog.
14. Click OK to close the *Map→2D Grid* dialog. This creates a new item in the Project Explorer under *Cartesian Grid Data* named “TUFLOW grid Grid”.
15. Right-click on “TULFOW grid Grid” and select **Rename** to enter a new name of “20m”.

## 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 reading 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 read in the area properties for this example and get the data into the map module:

1. Right-click on the tree item Map Data in the Project Explorer and select **New Coverage.** This brings up the *New Coverage* dialog.
2. Change the *Coverage Type* to “Area Property” under the Generic folder.
3. Enter *Coverage Name* of “materials”.
4. Click **OK** to exit the *New Coverage* dialog and create the new coverage.

5. Select the “materials” coverage (making it active). When converting GIS data to feature objects, the feature objects are added to the active coverage.
6. Select *File* | **Open** to bring up the **Open** dialog again.
7. Select the file “materials.shp” and click **Open**. This will load the data into the GIS module.
8. Select the “materials.shp” layer in the *GIS Data* folder in the Project Explorer.
9. From the *Mapping* menu choose **Shapes → Feature Objects**.
10. Click **Yes** to use all shapes to bring up the *GIS to Feature Objects Wizard*.
11. Choose “materials” under *Use an existing coverage* and click **Next**.
12. Under *MATNAME,* choose “Material” from the drop-down menu.
13. Click **Finish** to close the *GIS to Feature Objects Wizard.*

Notice that the area property coverage contains polygons 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:

1. Select *Edit* | **Materials Data** to bring up the *Materials Data* dialog.
2. Click the button **New**.
3. Double-click on the new material to rename the material. Enter “grasslands” as the new name.
4. 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. Create a boundary condition coverage in SMS by right-clicking on the folder Map Data and selecting **New Coverage** to bring up the *New Coverage* dialog again.
2. Change the *Coverage Type* to “1D–2D BCs and Links” under the TUFLOW folder and enter a *Coverage Name* of “BC.”
3. Click **OK** to close the *New Coverage* dialog then make sure the new “BC” coverage is active.
4. Using the **Create Feature Arc** tool, click out an arc at the location labeled “Upstream BC” in Figure 2.

Inflow boundary arcs should be created such that constant water surface (head) can be assumed along the arc. The arc as shown in Figure 2 is angled upstream in the floodplain as a better approximation of the correct equal head condition.

5. Select the newly created arc using the **Select Feature Arc** tool.
6. Right-click and select **Attributes** to bring up the *Boundary Conditions* dialog.
7. Change *Type* to “Flow vs Time (QT)”.
8. Click **Edit Events...** button to open the *TUFLOW BC Events* dialog.
9. Click the **Add** button and an event titled “new_event” will come up.
10. Double-click on “new_event” and change its name to “100 year”.
11. Click **OK** to close the *TUFLOW BC Events* dialog.
12. Select the “100 year” event.
13. Click on the box currently labeled **Curve undefined** to bring up the *XY Series Editor* dialog.
14. To copy the values needed for this tutorial:
    - Open the file “bc.xls” in a spreadsheet program (such as Microsoft Excel).
    - In the spreadsheet program, select cells *A2* through *B13* and copy them (*Ctrl-C*).
    - Select both columns in the *XY Series Editor* dialog by clicking in row 1 in the *Time (hrs)* column, then clicking in row 1 in the *Flow (cms)* column while holding down the *Ctrl* key.
    - Paste the contents previously copied (*Ctrl-V*). The values from the spreadsheet should now be listed in both columns, and the graph on the right will show a steep curve.
15. Click **OK** to close the *XY Series Editor* dialog.
16. Click **OK** to close the *Boundary Conditions* dialog.

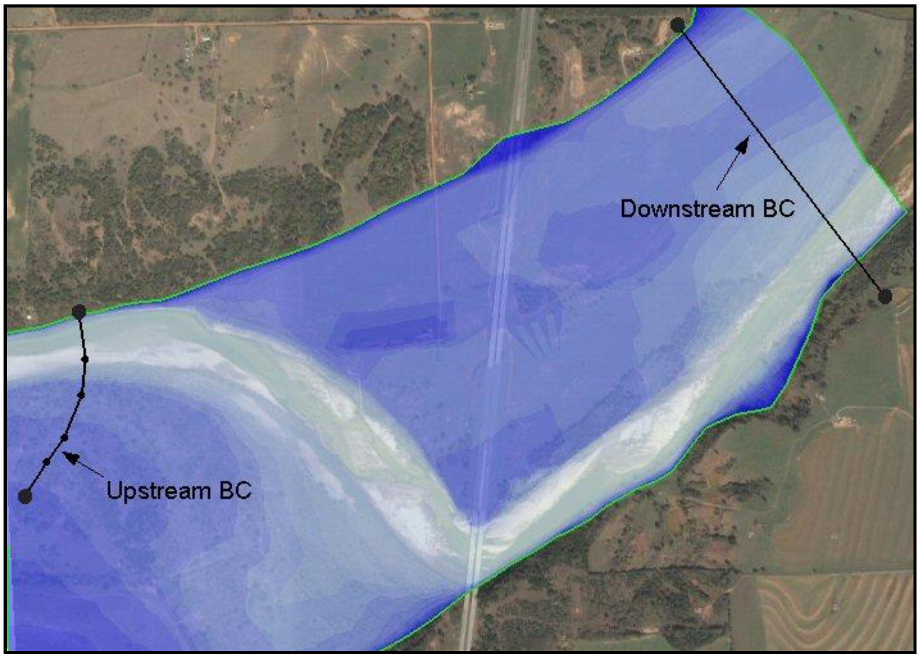



*Figure 2 Positions of Boundary Condition arcs*

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 (see Figure 2). 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. Set *Type* to be "Wse vs Time (HT)".
4. Select the "100 year" event and click on the box labeled **Curve undefined** which will bring up the *XY Series Editor* dialog.
5. To copy the values needed for this tutorial:
   - Open the file "bc.xls" in a spreadsheet program (if it is not still open).
   - In the spreadsheet program, select cells *A2* through *A13* and copy them (*Ctrl-C*).
   - In the *XY Series Editor* dialog, select row 1 in the *Time (hrs)* column and paste (*Ctrl-V*) the previously copied content.
   - In the spreadsheet program, select cells *C2* through *C13* and copy them.
   - Select row 1 in the *Wse (m)* column and paste (*Ctrl-V*) the previously copied content. The values from the spreadsheet should now be listed in both

columns, and the graph on the right will show a steep curve just like the one when the inflow boundary conditions were assigned .

6. Click **OK** to close the *XY Series Editor* dialog.
7. Click **OK** to close the *Boundary Conditions* dialog to return to the main screen in SMS.

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.

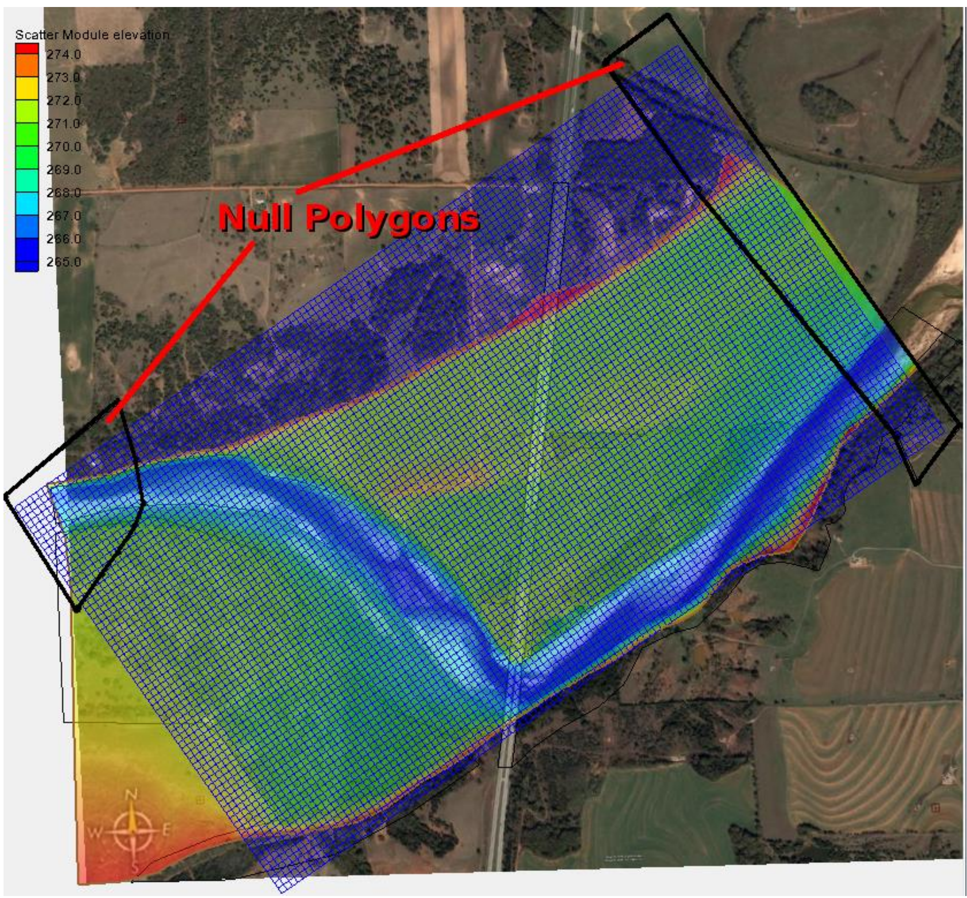



*Figure 3 Downstream inactive polygon*

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 loops around the entire domain

downstream of the arc and closes on the other end of the downstream boundary condition arc as shown in Figure 3.

2. Repeat this process to define a polygon on the upstream side of the upstream boundary condition as shown in Figure 3.

Enclosed arcs are not automatically converted into polygons in SMS. After creating enclosed arcs, build polygons by:

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

Next assign boundary conditions to the polygons by:

4. Select the **Select Feature Polygon** tool and double-click on the downstream polygon to bring up the *Boundary Conditions* dialog.
5. In the dialog, turn on *Set Cell Codes*.
6. Click the dropdown menu next to it and select the code to be "Inactive – not in mesh".
7. Click **OK** to close the *Boundary Conditions* dialog.
8. Repeat steps 4-7 for the upstream polygon.

# 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 each which includes "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 choose *New Simulation* | **TUFLOW**. This creates several new folders used throughout the rest of the tutorial. Under the tree item named "Simulations", there will be a new tree item named "Sim."
2. Right-click on the "Sim" tree item and select the command **Rename**. Give the simulation the name "100year_20m."

## 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 the folder named "Components" and choose **New 2D Geometry component**.
2. Right-click on the "2D Geom Component" tree item and select **Rename** then change the name to "20m_geo."
3. Drag the following Map Data coverages under the new "20m_geo" component:
   - "materials"
   - "BC"
4. Drag the "20m" grid under the "20m_geo" simulation 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:

1. Right-click on the "20m_geo" geometry component and select **Grid Options** to bring up the *Grid Options* dialog.
2. Under *Materials*, select the radio button "Specify using area property coverage(s)".
3. Change *Default material* to "grasslands."
4. Under *Cell codes* select the radio button "Specify using BC coverage(s)".
5. Change the default code to "water cell."
6. 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 the "Material Sets" folder and select **New Material Set**. A new "Material Set" will appear below the folder.
2. Right-click on the new "Material Set" and select **Properties** from the menu to bring up the *TUFLOW Material Properties* dialog. The materials are displayed in the list box in on the left.
3. Change the values for Mannings *n* (the field labelled "n") for the materials according to the table below:

| Material | Manning's *n* |
|---|---|
| Channel | 0.03 |
| Forest | 0.1 |
| Grasslands | 0.06 |
| Light forest | 0.08 |
| Roadway | 0.02 |

4. When finished, click **OK** to close the *TUFLOW Material Properties* dialog.

## 4.3 Simulation Setup and model parameters

The simulation includes a link to the geometry component as well as 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:

1. Drag the “20m_geo” geometry component onto the “100year_20m” simulation in the Project Explorer.
2. Right-click on the “100year_20m” simulation and select **2D Model Control** to open the *TUFLOW 2D Model Control* dialog.
3. Select the *Output Control* from the list on the left.
4. In the *Map Output* section, set:
    - *Format type* to “SMS 2dm”.
    - *Start Time* to “0” hours.
    - *Interval* to “900” seconds (15 minutes).
5. In the *Output Datasets* section, select the following datasets:
    - Depth.
    - Water Level.
    - Velocity Vectors.
    - Flow Vectors (unit flowrate).
6. In the *Screen/Log Output* section, change the *Display interval* to “6”. While TUFLOW is running, it will write status information every 6 time steps.
7. Select the *Time* from the list on the left.
8. Set the following:
    - *Start Time* to “2” hours.

- *End Time* to “16” hours.
- *Time step* to “5.0” seconds.

9. Select *Water Level* from the list on the left and change the *Initial Water Level* to “265.5”.
10. Turn on *Override default instability level (10m above highest elevation* and set *Instability level (m)* to “285.0”.
11. Select *BC* from the list on the left and switch the *BC event name* to “100 year”.
12. 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 a *Save* dialog.
2. Enter a *File name* of “Cimarron2d” and make certain the *Save as type* is set to “Project Files (*.sms)”.
3. Click the **Save** button to save the new project file.

# 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 the “100year_20m” simulation and select **Export TUFLOW files**. This will create 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 **Launch TUFLOW**. This will bring up a console window and launch TUFLOW. This process may 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 an *Open* dialog.
2. Select “100year_20m.tlf” in the *data files\TUFLOW\runs\log* directory and click **Open**.
3. A *View Data File* dialog may appear asking which program to use to open the file. Select “Notepad” or another text editor and click **OK**.
4. Scroll to the bottom of the file. The bottom of this file will report if the run finished, whether the simulation was stable, and report the number of warning and error messages. Some warnings and errors are found in the TLF file (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. Select *File* | **Open** to bring up the *Open* dialog again.
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. Under *Read As*, select “GIS layer” from the drop-down menu and click **OK** to close the *Mif/Mid import* dialog.
4. Nothing will happen because there are no errors in these files. 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. Select *File* | **Open** to bring up the *Open* dialog again.
2. Browse to the *data files\TUFLOW\results* folder.
3. Select “100year_20m.xmdf.sup” file and click **Open**.

The TUFLOW output is read into SMS in the form of a two-dimensional mesh. If a dialog pops up to replace existing material definitions, click no. If a dialog pops up and asks for time units, select hours.

4. From the Project Explorer, turn off all Map Data, Scatter Data, and Cartesian Grid Data.
5. Turn on and select Mesh Data to make it active.
6. Click on the **Display Options** button to bring up the *Display Option* dialog.
7. Select *2D Mesh* from the list on the left the turn on *Contours* and *Vectors* and turn off *Elements* and *Nodes*.
8. Switch to the *Contours* tab and select “Color Fill” as the *Contour method*.
9. Click **OK** to close the *Display Options* dialog.

The mesh will be contoured according to the selected dataset and time step.

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

# 9 Including the Roadway in the Model

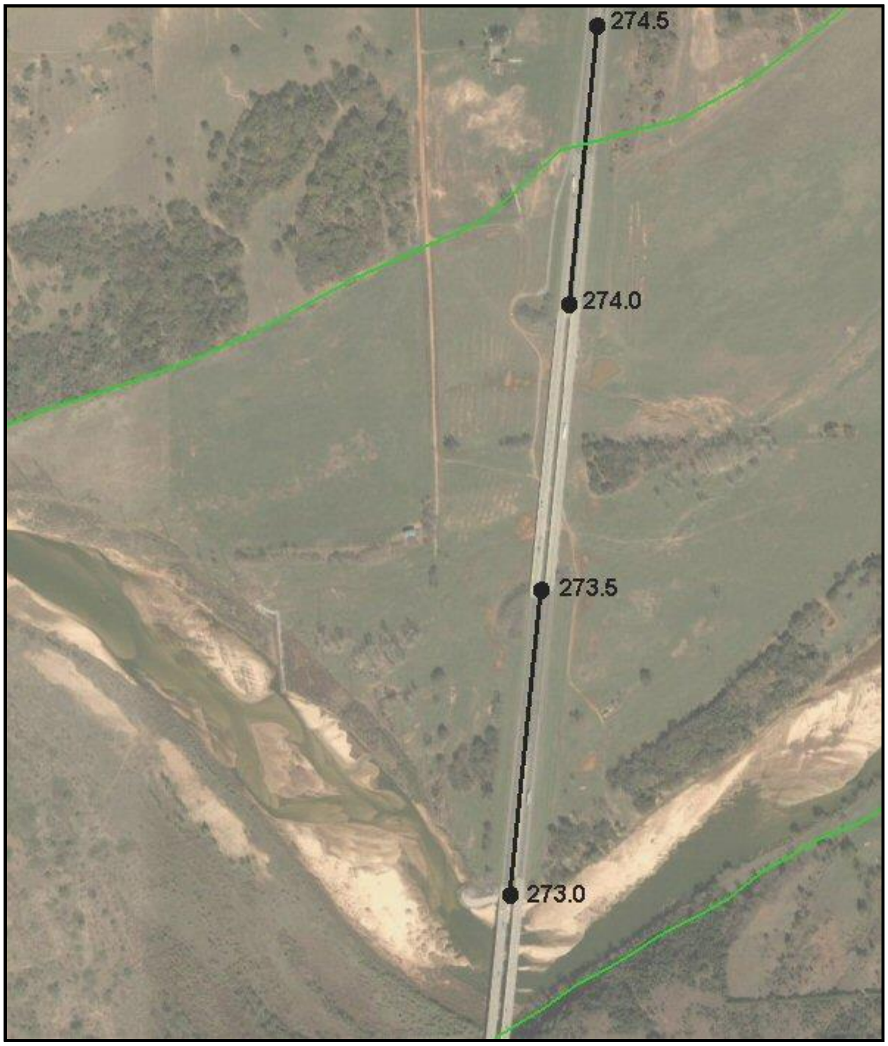



*Figure 4 Roadway embankment arc and elevations*

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:

1. Create a TUFLOW Geometry Modification by right-clicking on Map Data in the Project Explorer and selecting **New Coverage** to bring up the *New Coverage* dialog.
2. Set the *Coverage Type* to “1D–2D BCs and Links”, and enter a *Coverage Name* of “Roadway”.
3. Click **OK** to close the *New Coverage* dialog.
4. Turn off Mesh Data in the Project Explorer so the roadway is visible.
5. Using the **Create Feature Arc** tool, click out two arcs for the road embankments as shown in Figure 4.
6. Select the **Select Feature Point** tool. Change the elevation of each node to the appropriate value as shown in Figure 4 by selecting them and editing the Z options window on top of the screen.

# 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 the geometry component “20m_geo” and select **Duplicate**.
2. Right-click on the new component and select **Rename** then enter “20m_road” as the new name.
3. Drag the “Roadway” coverage onto the “20m_road” component.

Similarly, a new simulation needs to be created which uses this geometry component.

To create and setup the simulation:

1. Right-click on the simulation “100year_20m” and select **Duplicate**.
2. Right-click on the new simulation and select **Rename** then enter “100year_20m_road” as the new name.
3. Right-click on the grid component link in the “100year_20m_road” simulation labeled “20m” and select **Delete**. This deletes the link to the grid component, not the component itself.
4. 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. Enter a *File Name* of "Cimarron2d_road.sms" and click **Save**.
3. Right-click on the "100year_20m_road" simulation and select **Export TUFLOW files**.
4. Right-click on the "100year_20m_road" simulation and select **Launch TUFLOW**. This will bring up a console window and launch TUFLOW. This process may take several minutes to complete.
5. Click **OK** when the dialog indicates the simulation is finished.
6. Select *File* | **Open** to bring up the *Open* dialog again.
7. Browse to the *data files\TUFLOW\results* folder.
8. Select "100year_20m_road.xmdf.sup" file and click **Open**.
9. A *Select Tree Item for Datasets* dialog will appear asking for how to organize the solution datasets in the Project Explorer. This is because there are now multiple meshes as a result of the two model runs. Select "100year_20m_road" to place the dataset under this item then click **OK**.
10. Review the "100year_20m_road" solution datasets and click through the time steps to see the results.

## 12 Conclusion

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 above. A grid with 10 meter cells gives solutions without negative depth warnings.

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.