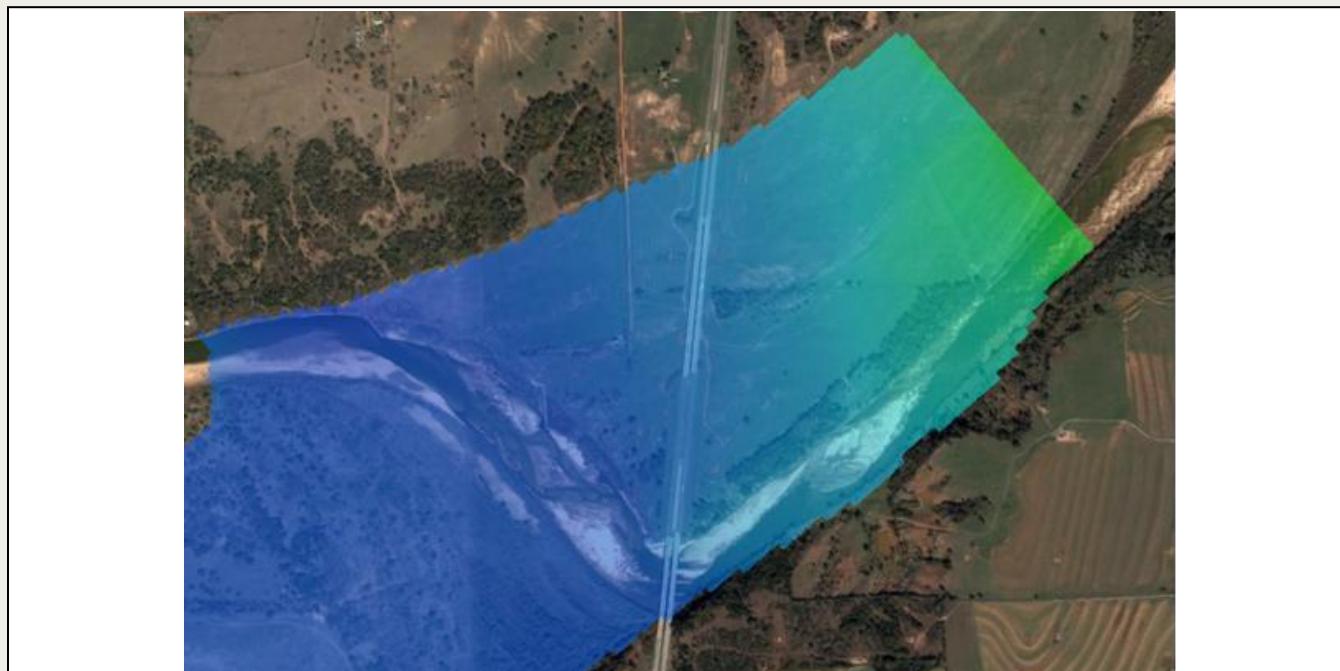




SMS 14.0 Tutorial

TUFLOW 2D Structures

Add structures to a TUFLOW model



Objectives

This tutorial describes using structures such as roadways in a TUFLOW project using the SMS interface.

Prerequisite Tutorials

- Overview
- TUFLOW 2D Hydrodynamics

Required Components

- SMS Core
- TUFLOW Model & Interface

Time

- 15–25 minutes

1	Introduction	2
1.1	Getting Started.....	2
1.2	Opening an Existing Project	2
2	Including a Roadway in the Model	3
2.1	Creating the Roadway Arc Coverage	3
2.2	Creating the Roadway Using Feature Objects	4
3	New Geometry Component and Simulation	4
4	Saving and Running the Simulation	5
5	Viewing the Results	5
6	Conclusion	6

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. The TUFLOW website contains more information regarding the model.¹

This tutorial discusses and demonstrates adding structures to a TUFLOW simulation. While it specifically discusses roads, the principles can be applied to any sort of structure. The area used in the tutorial is where I-35 crosses the Cimarron River in Oklahoma, about 50 miles north of Oklahoma City.

It is highly recommended that the “TUFLOW 2D Hydrodynamics” tutorial be completed prior to beginning this one. This tutorial uses the ending files from that tutorial.

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. Click the **Yes** button to clear all data in SMS.

1.2 Opening an Existing Project

Begin by opening an existing project that contains 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 the *File* | **Open...** menu command to bring up the *Open* dialog.
2. Browse to the SMS_TUFLOW_2D_Structures\data files folder and select “Cimarron2d.sms”.
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.

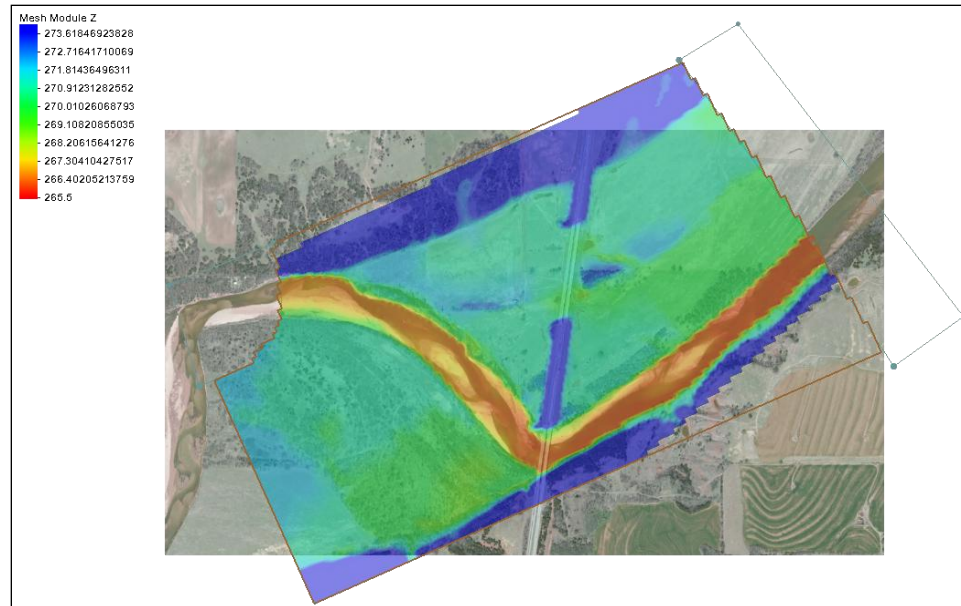


Figure 1 Contours with river portion showing yellow and orange

2 Including a Roadway in the Model

The bathymetry data in this project does 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.

2.1 Creating the Roadway Arc Coverage

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

1. 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 *Models | TUFLOW | 1D-2D BCs and Links*.
3. For the *Coverage Name*, enter "Roadway".
4. Click the **OK** button to close the *New Coverage* dialog.
5. Select the "Roadway" map coverage to make it active.

2.2 Creating the Roadway Using Feature Objects

Now to add feature objects to the coverage:

1. In the Project Explorer, turn off the “ Mesh Data” folder checkbox so the roadway is visible.
2. Using the **Create Feature Arc**  dynamic tool, click out two arcs for the road embankments as shown in Figure 2.
3. Using the **Select Feature Point**  dynamic tool, select the top node and enter “274.5” in the Z field at the top of the SMS window.
4. Repeat step 3 for each of the other three nodes, entering “274.0”, “273.5”, and “273.0”, as indicated in Figure 2.

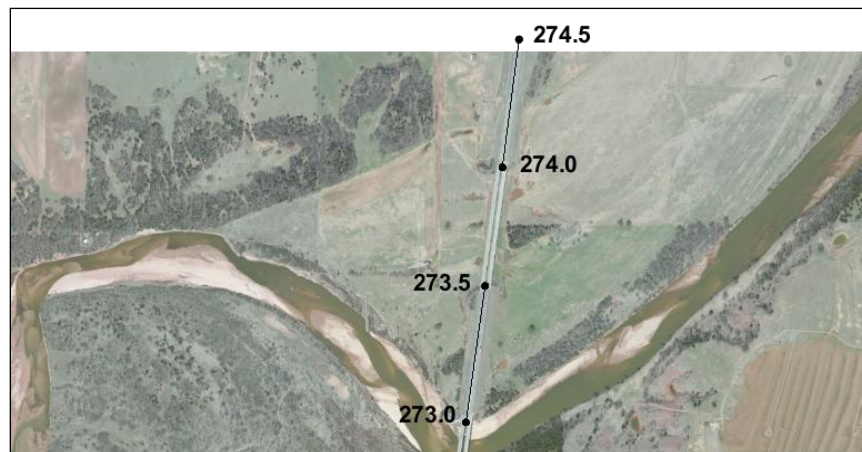


Figure 2 Roadway embankment arc and elevations

3 New Geometry Component and Simulation

Rather than changing the existing simulation, create a new simulation that includes the roadway. The ability to create additional simulations 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 “ 20m_geo” TUFLOW component and select the **Duplicate** context menu command.
2. Right-click on the new “ 20m_geo (2)” component and select the **Rename** context menu command.
3. Enter “20m_road” and press *Enter* to set the new name.
4. Drag the “ Roadway” map 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 the “ 100year_20m” simulation and select the **Duplicate** context menu command.

6. Right-click on the new “ 100year_20m (2)” simulation and select the **Rename** context menu command.
7. Enter “100year_20m_road” and press *Enter* to set the new name.
8. Right-click on the grid component link labeled “20m_geo” under the “ 100year_20m_road” simulation and select the **Delete** context menu command.
9. When asked to confirm, click the **Yes** button.

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

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

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

4 Saving and Running the Simulation

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_road.sms”.
4. Click the **Save** button to save the project under the new name and exit the *Save As* dialog.
5. Right-click on the “ 100year_20m_road” simulation and select the **Export TUFLOW files** context menu command.
6. Right-click on the “ 100year_20m_road” simulation and select the **Launch TUFLOW** context menu item 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.

5 Viewing the Results

Now to import the results and view them in 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_road.xmdf.sup”.
3. Click the **Open** button 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.

4. Select the “100year_20m_road” mesh.
5. Click the **OK** button to close the *Select Tree Item for Datasets* and to place the solutions under that mesh.
6. If prompted, specify the projection for the imported data.

7. Select the *Display* | **Display Options...** menu command to open the *Display Options* dialog.
8. From the list on the left, select *2D Mesh*.
9. From the tabs on the right, select *Vectors*.
10. In the *Vector display placement and filter* section, change the *X spacing* and *Y spacing* to “5” pixels.
11. Click the **OK** button to close the *Display Options* dialog.
12. Review the “100year_20m_road” solution datasets and click through the time steps to see the results.

The final time step, with “Water Level” and “Vector Velocity” mesh datasets selected, should appear similar to Figure 3.

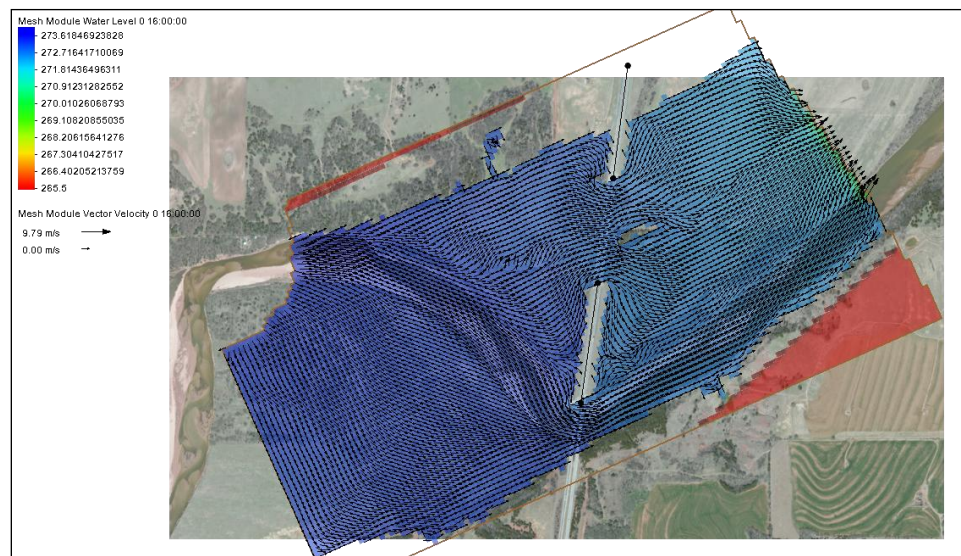


Figure 3 Final time step

The simulation message files may contain negative depth warnings, which indicate potential instabilities. These can be reduced by increasing the resolution of the grid and decreasing the time step as required.

Please refer to the steps in the “TUFLOW 2D Hydrodynamics” tutorial for more details regarding viewing message file information. A grid with 10 meter cells gives solutions without negative depth warnings.

These steps can be used to add a variety of different structures to the simulation, including walls, piers, and so on.

6 Conclusion

This concludes the “TUFLOW 2D Structures” tutorial. Feel free to continue experimenting with the effects of changing material properties, if desired. 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.