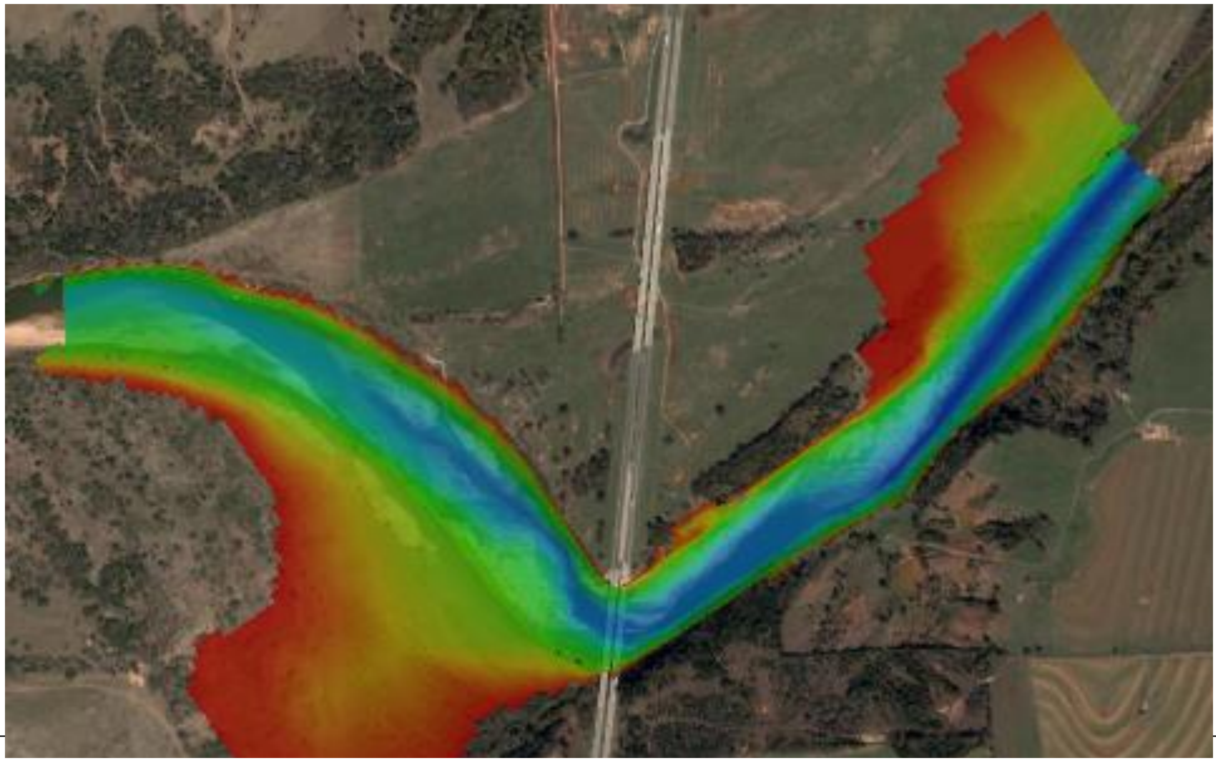
SMS

v. 12.3

SMS 12.3 Tutorial

TUFLOW 1D/2D



Objectives

This tutorial describes the generation of a 1D TUFLOW project using the SMS interface. It is strongly recommended that the TUFLOW 2D tutorial be completed before doing this tutorial.

Prerequisites

- TUFLOW 2D Tutorial

Requirements

- Map Module
- Grid Module
- Scatter Module
- Grid Module

Time

- 75–120 minutes

AQUAVEO™

1 Introduction and Background Data

TUFLOW is a hydraulic model that can work with mixed 1D/2D solutions. It handles wetting and drying in a very stable manner. More information about TUFLOW can be obtained from the TUFLOW website.[1]

This tutorial focuses on adding 1D cross sections to a 2D model of where I-35 crosses the Cimarron River in Oklahoma, about 50 miles north of Oklahoma City. It starts with the grid as created in the 2D TUFLOW tutorial. Refer to that tutorial to learn how to setup the grid.

The modeling process when using combined 1D and 2D components with the TUFLOW model includes the following:

- Defining the 2D domain or active portion of the grid.
- Specifying the 1D network (center line and cross sections).
- Defining the 1D/2D connections
- Specifying the boundary conditions
- Combining all the components into a simulation.

A TUFLOW model uses grids to define the two dimensional (Eulerian) domain. It uses GIS objects grouped into feature coverages to define modifications to the grid such as levies or embankments. Feature objects can also be used to define additional objects

[1] See http://www.tuflow.com/

such as cross sections and channel centerlines. A TUFLOW simulation consists of a group of these geometrical objects, along with model parameters and specifications. Note that all units in TUFLOW must be metric.

In this tutorial, a model will be built that uses a 1D cross section based solution within the channel and 2D cell-based solution outside the channel. A 1D/2D model gives better channel definition than an all-2D model because the cross sections have higher resolution than the 2D grid allows. Additionally, a 1D/2D model generally has a shorter computation time.

To start the tutorial:

1. Click *File* | **Open…** to bring up the *Open* dialog.
2. Select "Project Files (*.sms)" from the *Files of type* drop-down.
3. Browse to the *data files* folder for this tutorial and select "Cimmaron_1D.sms".
4. Click **Open** to import the project and exit the *Open* dialog.

This imports an SMS project with a background image, elevation data, the 20m grid created in the 2D tutorial, and three map coverages. The project should appear similar to Figure 1.

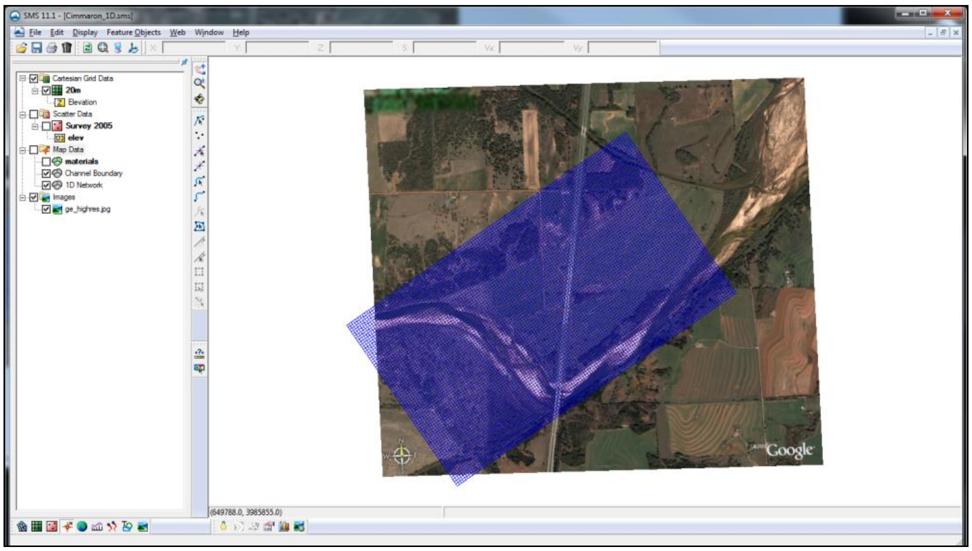


Figure 1 View of Map and Image Data

2 1D/2D TUFLOW models

TUFLOW supports several methods for linking 1D and 2D models as described in the TUFLOW reference manual, including:

- Embedding a 2D domain inside a large 1D domain (Figure 2-1a).
- Insert 1D networks "underneath" a 2D domain (Figure 2-1b, and Figure 3).
- Replace, or "carve" a 1D channel through a 2D domain (see Figure 2-1c, and Figure 4).

This tutorial illustrates the third method.

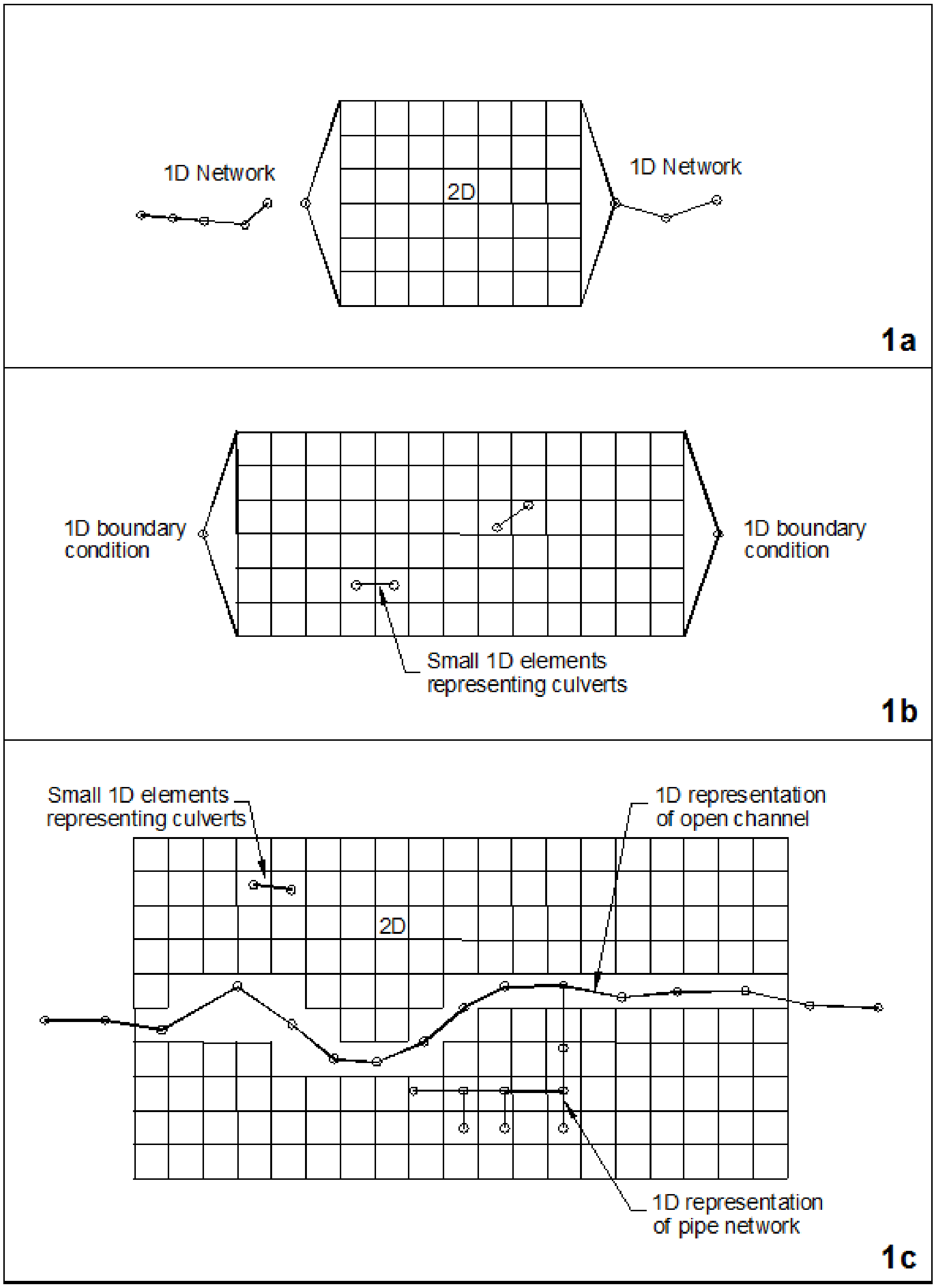


Figure 2 1D/2D linking mechanisms[2]

[2] *TUFLOW User's Manual, 2010* (Build 2010-10-AB), p.3–4 "The Modelling Process".

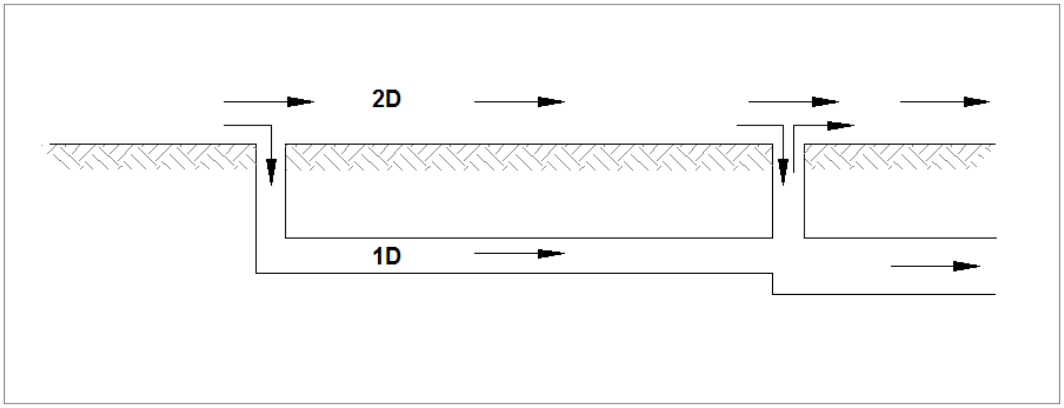


Figure 3 Modelling a pipe system in 1D underneath a 2D domain[3]

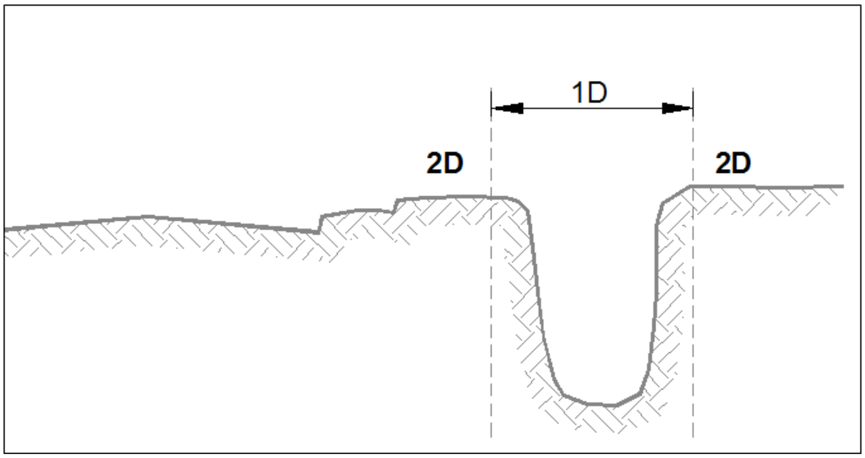


Figure 4 Modelling a channel in 1D and the floodplain in 2D[4]

3 Defining the 2D Portion of the Model

The TUFLOW 2D tutorial demonstrated how to define a grid to represent the geographical features— such as a flood plain— in a study domain. The combined 1D/2D simulation continues to utilize this grid. However, portions of the grid are disabled or eliminated because that portion of the simulation will be represented with a 1D network. These portions need to be defined.

3.1 2D Computation Domain

For this tutorial, the region that should not be included in the 2D calculations has already been defined in one of the coverages.

1. Select " Channel Boundary" to display this polygon.

[3] Ibid, pp.3–5.

[4] Ibid.

Note that the orange polygon encloses the channel and the regions both upstream and downstream from the study area. The area in this polygon will be simulated using 1D analysis.

To specify that this polygon is not to be included in the 2D calculations, assign an attribute using these steps:

2. Right-click on “ Channel Boundary” and select *Type* | *Models* | *TUFLOW* | **1D/2D BCs and Links**.
3. Using the **Select Feature Polygon** tool, double-click inside the channel to bring up the *Boundary Conditions* dialog.
4. Select “No BC” from the *Type* drop-down.
5. In the *Options* section, turn on *Set cell code* and select “Inactive -- not in mesh” from the drop-down.

When using this option, TUFLOW will not create 2D cells in this area.

6. Click **OK** to exit the *Boundary Conditions* dialog

4 Setting up the 1D Network

Several coverages are used to define the 1D cross section based network. The first coverage created will be of type “TUFLOW 1D Network.” This coverage will be used to define the centerline for the channels as well as the attributes for the weir. For this tutorial, the center channel points have already been given. In other projects, these points would normally be created manually with guidance from the underlying image or topographic data.

To create the channels:

1. Turn off " 20m" in the Project Explorer to reduce the amount of data visible on the screen.
2. Select “ 1D Network” to make it active.
3. Right-click on “ 1D Network” and select *Type* | *Models* | *TUFLOW* | **1D Networks**.
4. Using the **Create Feature Arc** tool, create a series of arcs (one for each node pair) to connect the nodes from left to right, starting and stopping at each node.

By default, each arc represents a segment of open channel, with the length coming from the channel it is representing. Each arc should represent a fairly consistent cross section shape. Intermediate vertices may be added as desired to make the centerline smoother. The finished digitized arcs should appear similar to the one in Figure 5.

With the channel centerline defined, set the most upstream arc as a weir. A wide weir will get the flow into the model and spread the flow downstream into both the 1D and 2D domains.

5. Using the **Select Feature Arc** tool, double-click the leftmost arc in the network to bring up the *Channel Attributes* dialog.
6. Select “Weir” from the *Type* drop-down.

7. Click **Attributes** to bring up the *Weir Attributes* dialog.
8. In the Geometry section, select *Define rectangular section*.
9. Enter “264.0” as the *Invert?*.

This is the elevation of the channel.

10. Enter “1000.0” as the *Width* (wide enough to cover the majority of the floodplain).
11. Click **OK** to exit the *Weir Attributes* dialog.
12. Click **OK** to exit the *Channel Attributes* dialog.

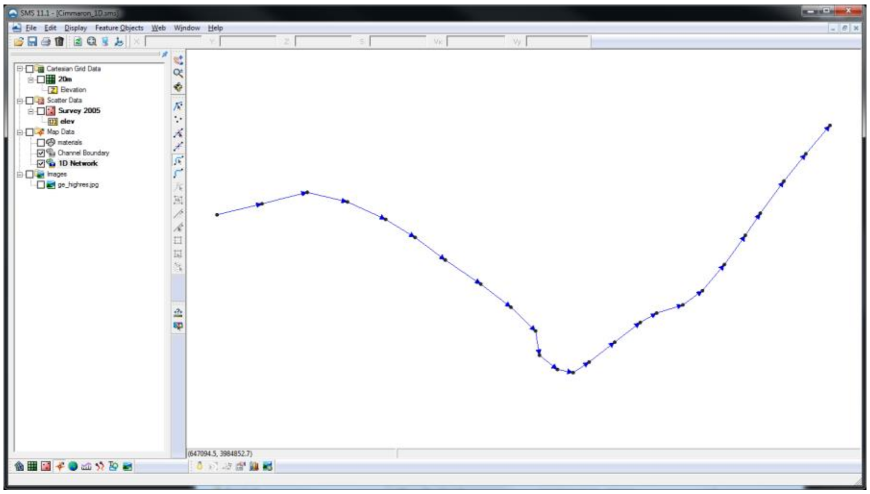


Figure 5 Creating the 1D network centerline

4.1 Creating Cross Sections

Each open channel arc uses cross section geometry to compute hydraulic properties (such as area and wetted perimeter) for each channel segment. TUFLOW needs to have a geometric definition of the channel for each segment as well as invert elevations at the cross section end points. The invert elevations define channel slope. Cross sections can be defined in the middle of a channel, at the channel endpoints, or both.

If cross sections are specified at the endpoints, the cross section information used for each channel is averaged from the cross section at each end. TUFLOW then extracts the channel inverts from the cross section definitions. If cross sections are specified at the middle of the channel segment, the upstream and downstream inverts must be specified manually.

If cross sections exist at both the endpoints and within the channel, the cross section properties are taken from the cross section within the channel and the inverts from the cross sections at the ends. For this tutorial, cross sections will be created at the end of each channel.

Cross sections will be laid out from the channel segments in the network coverage and then trimmed to the edge of the 1D domain, all using tools in SMS. Once the cross sections have been defined, extract elevations for them from the elevation data in the TIN and material data from the area property coverage.

To layout and trim the cross sections.

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 Cross Sections**.
3. Enter "Cross Sections" as the *Coverage Name*.
4. Click **OK** to close the *New Coverage* dialog and bring up the *CsDb Management* dialog.
5. Click **OK** to close the *CsDb Management* dialog as the cross section data will be added later.
6. Click **Display Options** to bring up the *Display Options* dialog.
7. Select "Map" from the list on the left.
8. Turn on *Inactive coverage* and click **OK** to close the *Display Options* dialog.
9. To make the 1D network more visible, turn off " materials", " Cartesian Grid Data", " Survey 2005" , and " ge_highres.jpg".
10. Right-click on " 1D Network" and select **Create Cross Section Arcs** to bring up the *Create Cross Section Arcs* dialog.
11. In the Extract locations section, turn off *Midpoints*.
12. In the *Cross section lengths* section, enter "350.0".

This ensures that the cross sections cover the entire 1D domain. The excess will be trimmed off later.

13. Click **OK** to close the *Create Cross Section Arcs* dialog.

This creates cross sections at the endpoint of each arc perpendicular to the arc (or arcs if it meets another arc). The cross sections are all 350 meters long. This method can create a lot of cross sections quickly but some manual cleanup will be necessary.

The goal is to have cross sections that are basically perpendicular to the channel. Local meanders in the channel arcs can cause cross section orientation to change. For this reason, representing excessive meanders in the channel sections is not recommended.

To manually cleanup the cross sections:

1. Using the **Select Feature Point** tool, move nodes on the inside bank of the river bend near the middle of the project so the cross section arcs do not overlap.

The cross sections do not need to be completely straight (one node can be moved without moving the opposite node on the cross section).

2. Using the **Select Feature Arc** tool, select the leftmost cross section (the first one) and *Delete* it.
3. Click **Yes** when asked to confirm the deletion.

This node is the weir, so the cross section at this location is not needed.

4. Using the **Select Feature Point** tool, drag the nodes of the first and last cross sections so they snap to the nodes on the channel boundary coverage.

This connects the first and the last cross sections with the extents of the 2D domain. All of the cross sections generated (except for those moved already) extend outside of the 1D channel boundary. The cross sections within the channel area are the only ones required here.

To trim the cross sections to the boundary:

5. Right-click on " Cross Sections" and select **Trim to code polygon**.

This trims the cross sections to the code polygons in the boundary condition coverage. Since there is only one boundary condition coverage, it will be used automatically. When working with a project that uses more than one such coverage, a dialog appears that allows choosing a coverage.

When done, the cross sections should appear similar Figure 6.

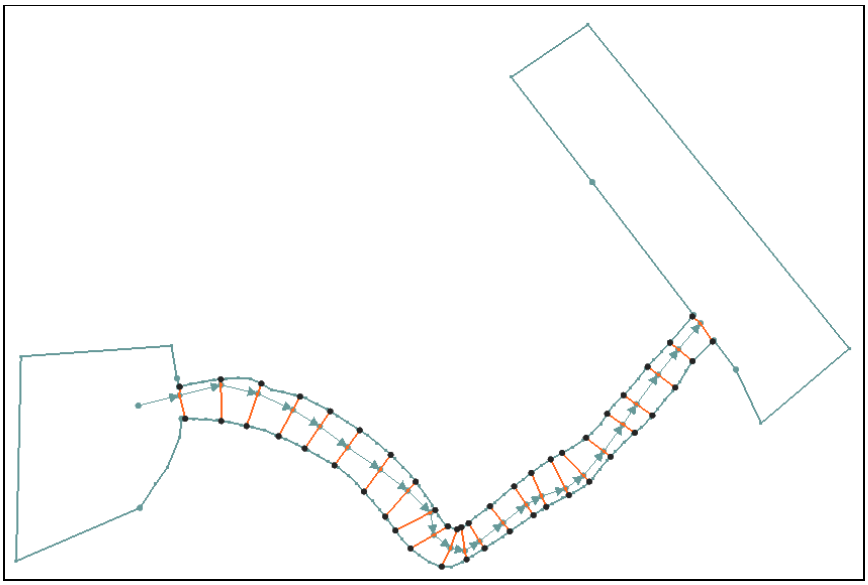


Figure 6 Final trimmed cross section arcs

With the cross sections laid out and trimmed, extract elevation and material data by doing the following:

1. Right-click on " Cross Sections" and select **Extract from Scatter**.

This extracts elevation data from the active dataset in the active scatter set (TIN), though there will be no visible changes on the screen.

2. Right-click on " Cross Sections" and select **Map Materials From Area Coverage** to bring up the *Select Coverage* dialog.
3. Select " materials".
4. Select "channel" from the *Default material* drop-down.
5. Click **OK** to close the *Select Coverage* dialog.

The cross sections now have elevation and material information. The data used for each cross section can be viewed or edited by doing the following:

6. Using the **Select Feature Arc** tool, double-click on one of the cross section arcs to bring up the *TUFLOW Cross Section* dialog. Notice that the cross section ID is shown in this dialog.

7. Click **Edit** to bring up the *Cross Section Attributes* dialog.

This dialog includes a plot of the cross section with several tools to edit the cross section data (Figure 7).

On the *Geom Edit* tab, the coordinates which define the cross section can be edited. The edits can be done graphically in the plot or by editing the spreadsheet. The *x* and *y* coordinates represent the location of the cross section in plan view and are ignored by TUFLOW. The *d* value is the distance along the cross section from the left bank toward the right bank. The *z* value is the elevation of the point.

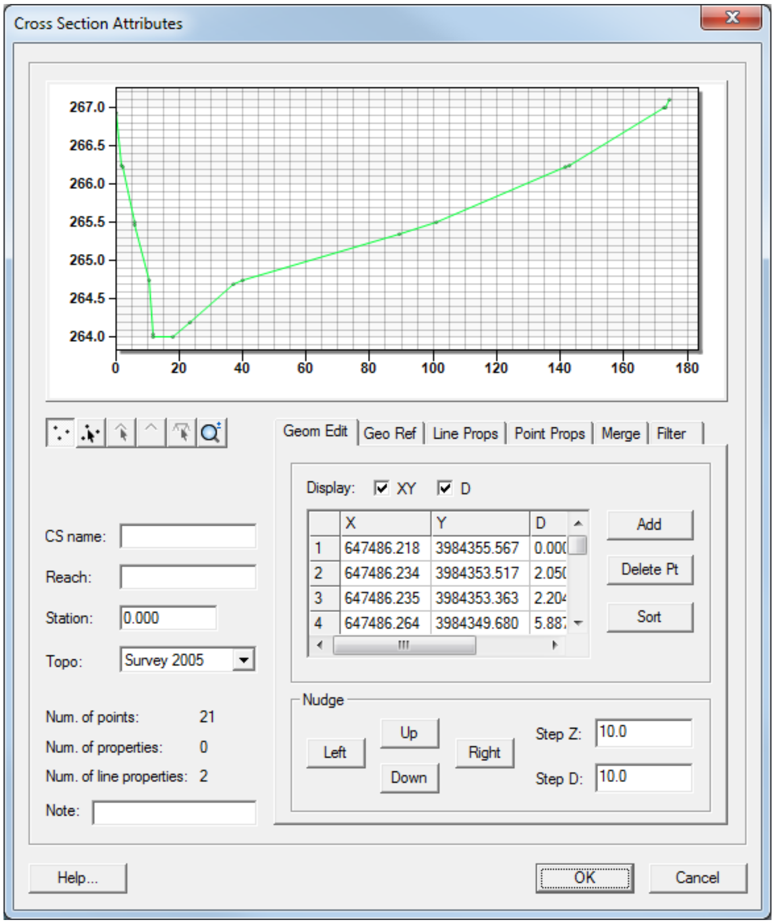


Figure 7 Cross Section Attributes dialog

The *Line Props* tab shows the materials that are assigned to each segment of the cross section. The material breaks may be edited in this dialog using the tools in the plot window or the spreadsheet below it. Additional information can be incorporated into the cross section here.

The other tabs (*Geo Ref*, *Point Props*, *Merge*, and *Filter*) are not used in this tutorial. They may be explored as desired at a later time.

8. Since there is no additional data, click **Cancel** to close the *Cross Section Attributes* dialog.
9. Click **Cancel** to close the *TUFLOW Cross Section* dialog.

Another useful tool to see cross sections is the TUFLOW cross section plot. With this tool several different cross sections can be selected and viewed at the same time.

10. Select *Display* | **Plot Wizard…** to bring up the *Step 1 of 2* page of the *Plot Wizard* dialog.
11. In the *Plot Type* section, select "TUFLOW Cross Section" from the list on the left and click **Finish** to close the *Plot Wizard* dialog.

The Main Graphics Window will split into a "Plot 1" window at the top and a "Cimmaron_1D.sms" window at the bottom.

12. Using the **Select Feature Arc** tool, select any one of the cross sections to display the cross section in the "Plot 1" window.
13. Holding *Shift* while using the **Select Feature Arc** tool allows selection and comparison of multiple cross sections.

The last arc selected will be in blue, while the other arcs will be in green (Figure 8).

14. Close the *Plot 1* window when done reviewing the plots by clicking on the **Close** button at the upper right corner of the *Plot 1* window (not the main SMS window).
15. If desired, expand the *Cimmaron_1D.sms* window by clicking on the **Maximize** button at the upper right corner of the *Cimmaron_1D.sms* window in the Main Graphics Window.

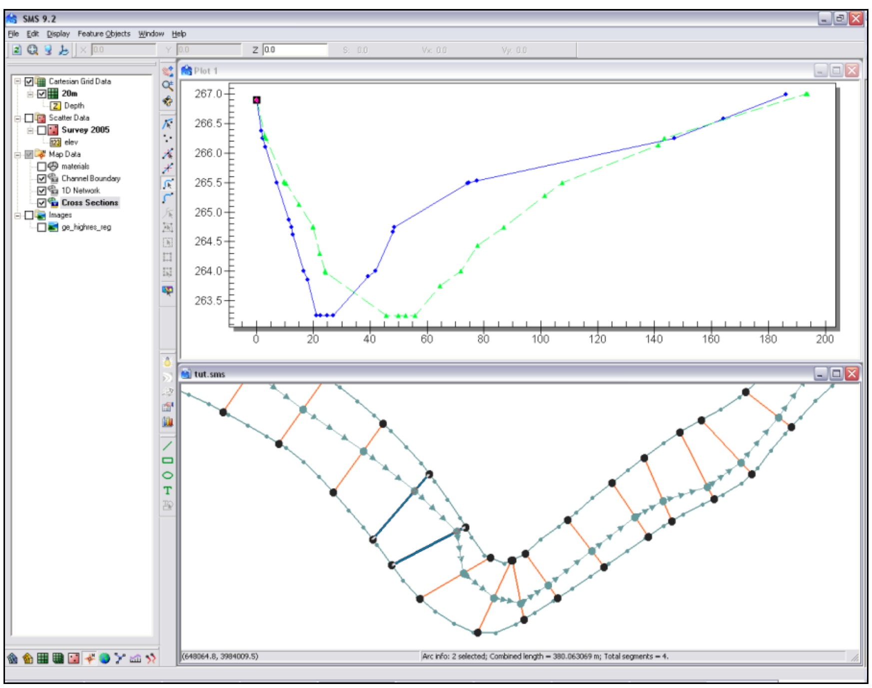


Figure 8 TUFLOW cross section plot

5 Defining the 1D/2D Connection

It is necessary to tell TUFLOW where flow will be allowed to move between the 1D and 2D domains. The main flow exchanges will be along both banks of the channel. At the top of the model, all flow will enter a wide 1D domain and then the flow will be split into the 1D domain for the channel flow and into the 2D domain for the floodplain flow.

5.1 1D/2D Flow Interfaces

In the SMS interface the locations for flow exchange are called “1D flow/2D water level (HX)” connections or sometimes as “HX Lines”. In this tutorial, “HX arcs” will be used to refer to these locations as these locations will be represented by feature arcs.

The first type of location where this transition will take place is on both sides of the channel and at the upstream end of the model where flows will change from 1D to 2D. The downstream end will be handled differently and will be discussed later.

To define these locations:

1. Select " Channel Boundary" to make it active.
2. Using the **Select Feature Arc** tool and while holding down the *Shift* key, select the arcs running along both banks of the channel and the single arc on each side of the upstream (left) end of the model (Figure 9)

The arc on the top left side of the channel is very short because the terrain in that region is very steep, and flow will not enter the domain beyond the extent of that arc.

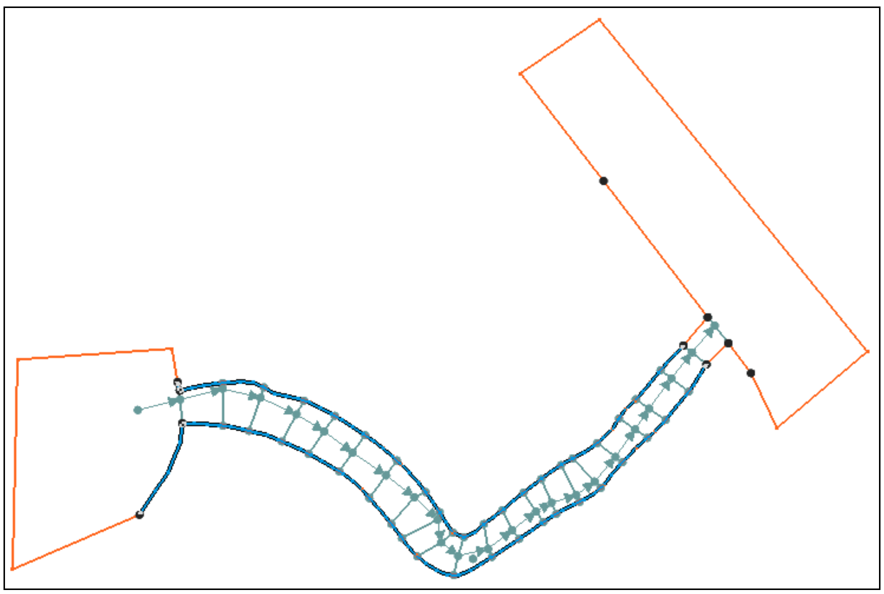


Figure 9 Selected HX arcs

3. Right-click and select **Attributes** to bring up the *Boundary Conditions* dialog.
4. Select “1D Flow/2D Water Level Connection (HX)” from the *Type* drop-down.
5. Click **OK** to close the *Boundary Conditions* dialog.

5.2 1D/2D Connections

There are two parts to defining 1D/2D links: on the 2D domain and on the 1D domain. The flow interfaces between the 1D and 2D domains in the "Channel Boundary" coverage are already defined. TUFLOW associates these arcs with the 2D domain spatially.

Along with these HX arcs, the 1D/2D connection from the 1D network nodes must be defined. These 1D/2D connection arcs tell TUFLOW which locations along the HX arcs match individual nodes.

Since the cross section arcs are in the same place as the placement locations of the 1D/2D connections, start with a copy of the cross section coverage:

1. Right-click on " Cross Sections" coverage and select **Duplicate** to create coverage named " Cross Sections (2)".
2. Right-click on " Cross Sections (2)" and select **Rename**.
3. Enter "1D_2D_Connection" and press *Enter* to set the new name.
4. Right-click on " 1D_2D_Connection" and select *Type* | *Models* | *TUFLOW* | **1D–2D Connections**.
5. Select " 1D_2D_Connection" to make it active.
6. Using the **Select Feature Vertex** tool, right-click in a blank area and choose **Select All**.

This selects all of the vertices. There is one vertex at the center of each cross section.

7. Right-click again and select **Convert to Nodes**.

This splits each of the arcs into two arcs, creating separate arcs connecting each node on the centerline to the HX arcs.

8. Right-click on " 1D_2D_Connection" and select **Properties…** to bring up the *Select Boundary Condition Coverage* dialog.
9. Select " Channel Boundary" in the tree list.
10. Click **OK** to close the *Select Boundary Condition Coverage* dialog.

TUFLOW requires that the HX arcs (in the " Channel Boundary" coverage) have a vertex at each 1D/2D connection point. SMS can enforce this.

11. Right-click on " 1D_2D_Connection" and select **Clean Connections** to bring up the *Clean Options* dialog.
12. Enter "5.0" as the *Tolerance*.
13. Turn on *2D BC Coverage (HX Lines)*.
14. Select "Channel Boundary" in the first tree list.
15. Click **OK** to close the *Clean Options* dialog.

This makes sure that connections arcs end at HX boundaries and the HX boundaries have vertices at the connection endpoints. Although there is no visible change in the screen, new vertices will be created, and if needed, the arc end points will be moved.

Connections have now been created from all of the 1D nodes on the centerline to the HX arcs that are mapped to the 2D domain. The 1D/2D connection needs to be defined at the upstream end of the domain. This is done by connecting the two HX arcs at the extreme end points to the 1D network. Since the most upstream channel segment was changed to a weir, the downstream node of that segment can be connected to these HX arc end points to define the transfer.

To do this:

16. Select "1D_2D_Connection" to make it active.
17. Using the **Create Feature Arc** tool, and create two arcs connecting the upstream side of the 2D domain to the downstream node of the 1D weir boundary (Figure 10).

The length or shapes of the 1D/2D connection arcs do not matter. TUFLOW simply uses their end points to connect nodes in the 1D network with cells in the 2D grid.

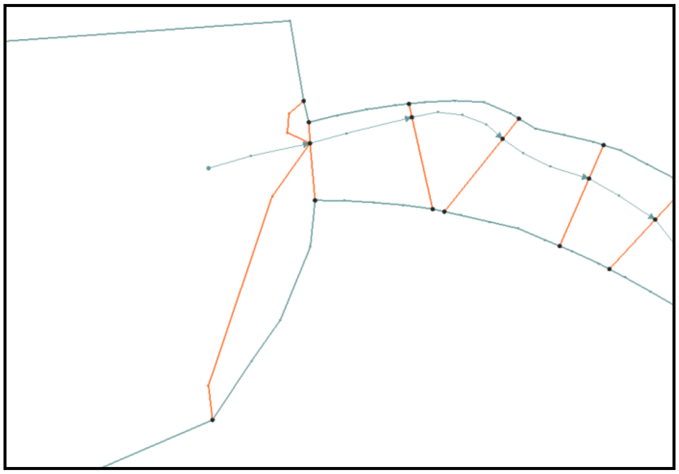


Figure 10 1D/2D Connection Arcs added at upstream end.

The downstream end of the domain needs to be cleaned up as well. It is possible to transition all the flow back into the 1D network by defining the 1D network past the 2D domain.

In this example, both 1D and 2D boundary conditions have been assigned on the downstream end of the model. This eliminates the need to extend the 1D network and define the connections, but it requires some interaction.

There cannot be any 1D/2D connection at the location of the boundary condition. To remove them:

18. Using the **Select Feature Arc** tool, select the two most downstream 1D/2D connection arcs (Figure 11) and press *Delete*.
19. Click **Yes** when asked to confirm the deletion.

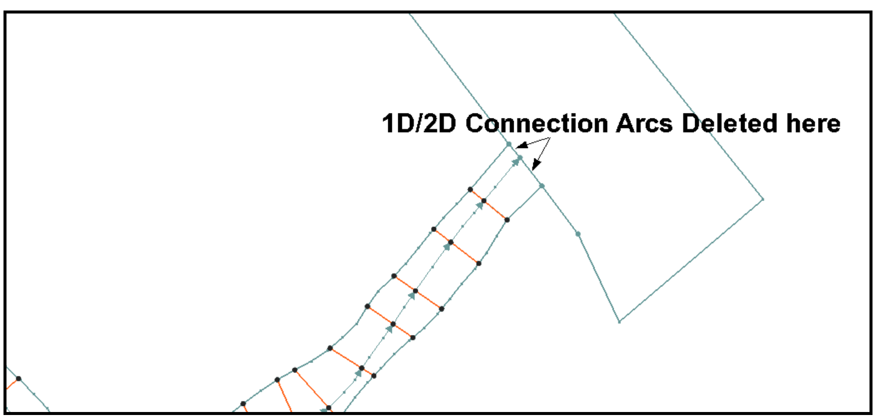


Figure 11 Location of deleted 1D/2D connection arcs on downstream end.

If examining the HX arcs created in the previous section, note that they end one cross section above the end of the 1D network. This was done to prevent an illegal 1D/2D connection at the boundary condition.

The connection arcs just deleted actually didn't connect to any HX arcs for this reason. This is a limitation of the 1D/2D boundary condition. There will be no transfer of flow between the 1D network and the 2D grid in this last channel segment.

20. Select " Channel Boundary" to make it active.
21. Using the **Select Feature Arc** tool, double-click on the most upstream arc (Figure 12) to bring up a *Boundary Conditions* dialog.
22. Select "1D Flow/2D Water Level Connection (HX)" from the *Type* drop-down.
23. Click **OK** to close the *Boundary Conditions* dialog.

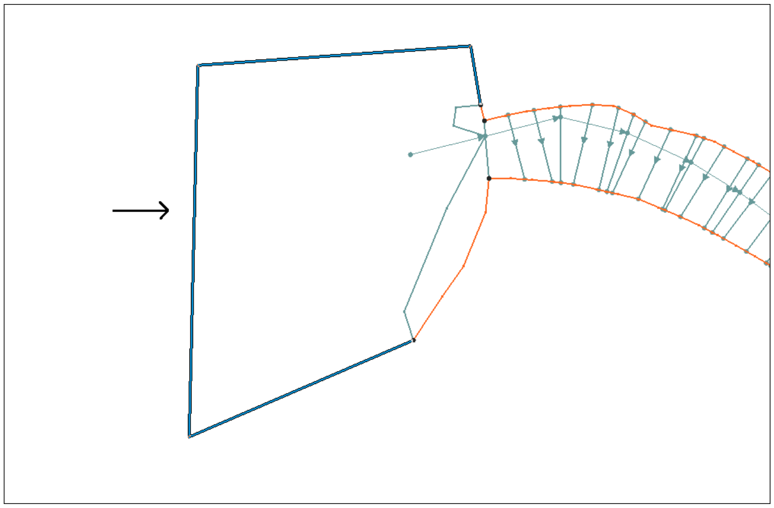


Figure 12 Channel boundary arc to select

6 Specifying the boundary conditions

As with any numerical model, it is necessary to specify the boundary conditions. This principally defines where the flow enters and leaves the simulation. This tutorial will have flow enter the simulation in the 1D network, and leave both the 1D network and the 2D grid. The inflow will be specified as a flow rate over the weir. The outflow will be controlled by specifying a head condition.

6.1 2D Downstream Water Level Boundary Condition

On the downstream end of the domain, it will be necessary to assign a water level boundary condition to both the 1D domain and to the 2D domain.

Since the 2D domain is split by the 1D domain, there will be two water level boundary condition arcs in this coverage.

1. Select " Channel Boundary" to make it active.
2. Using the **Select Feature Arc** tool while pressing the *Shift* key, select the two arcs along the downstream side of the 2D domain as shown in Figure 13.

3. Right-click and select **Attributes…** to bring up the *Boundary Conditions* dialog.
4. Select “Wse vs Time (HT)” from the *Type* drop-down.
5. In the *Events* section, click **Edit Events…** to bring up the *TUFLOW BC Events* dialog.
6. Click the **Add** button to create a new event named “new_event”.
7. Double-click on “new_event” and enter “100 year”.
8. Click **OK** to close the *TUFLOW BC Events* dialog.
9. Select “100 year” and click **Curve undefined** to bring up the *XY Series Editor* dialog.
10. Outside SMS, browse to the *data files* folder for this tutorial and open the “bc.xls” file in a spreadsheet editor.
11. Copy the values from the *Time* column in the spreadsheet and paste them into the *Time (hrs)* column in the *XY Series Editor* dialog.
12. Copy the values from the *Head (m)* column in the spreadsheet and paste them into the *Wse (m)* column in the *XY Series Editor* dialog.
13. Click **OK** to close the *XY Series Editor* dialog.
14. Click **OK** to close the *Boundary Conditions* dialog.

Rename the downstream arcs (Figure 13) by doing the following:

15. Using the **Select Feature Arc** tool, double-click on the arc to the left of the channel to bring up the *Boundary Conditions* dialog.
16. In the *Options* section, turn on *Override default name* and enter “downstream_wl_left” in the field to the right.
17. Click **OK** to close the *Boundary Conditions* dialog.
18. Repeat steps 15-17 for the arc to the right, entering “downstream_wl_right” as the name.

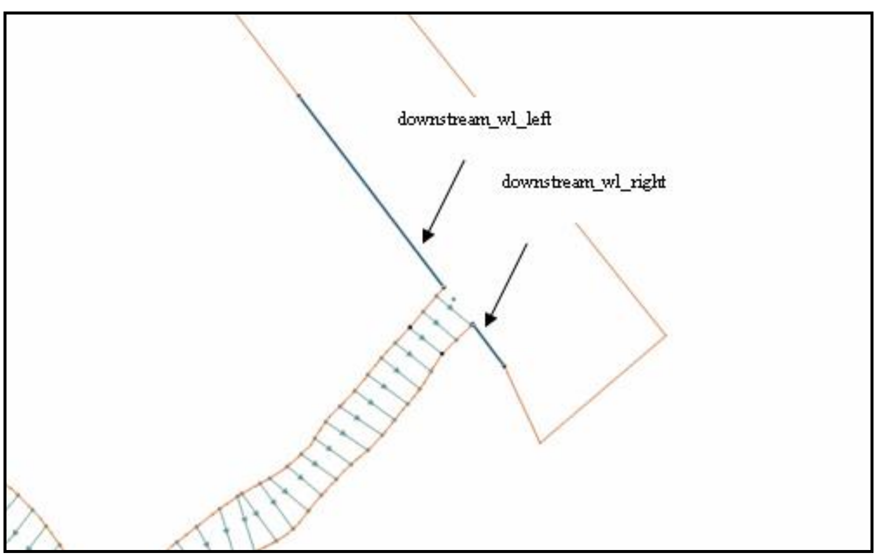


Figure 13 Location of downstream boundary condition arcs

6.2 Creating the 1D BC

Define the 1D network boundary conditions for both the upstream and downstream boundaries by doing the following:

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 BC and Links**.
3. Enter "1d_bc" as the *Coverage Name* and click **OK** to close *New Coverage* dialog.
4. Turn off " Channel Boundary", " 1D_2D_Connection", and " Cross Sections" to make it easier to see the 1D network.
5. Select " 1d_bc" to make it active.
6. Using the **Create Feature Point** tool, create points directly on top of the first and last nodes in the network coverage.

When getting close to the nodes in the other coverage, there should appear red crosshairs. This indicates that the node will snap to the existing node in the other coverage. If the red crosshairs do not appear, hitting *S* on the keyboard will activate this snapping functionality.

7. Using the **Select Feature Point** tool, double-click the upstream node to bring up the *Boundary Conditions* dialog.
8. Select "Flow vs Time (QT)" from the *Type* drop-down.
9. In the *Options* section, turn on *Override default name* checkbox and type "Upstream_1D".
10. In the *Events* section, select "100 year" and click **Curve undefined** to bring up the *XY Series Editor* dialog.
11. Outside of SMS, browse to the *data files* folder for this tutorial and open "bc.xls" into a spreadsheet program (if it is not already open).
12. Copy the values from the *Time* column in the "bc.xls" spreadsheet and paste them into *Time (hrs)* column in the *XY Series Editor* dialog.
13. Copy the values from the *Inflow (cms)* column in the "bc.xls" spreadsheet and paste them into *Flow (cms)* column in the *XY Series Editor* dialog.
14. Click **OK** to close the *XY Series Editor* dialog.
15. Click **OK** to close the *Boundary Conditions* dialog.
16. Using the **Select Feature Point** tool, double-click the downstream node to open the *Boundary Conditions* dialog.
17. Select "Wse vs Time (HT)" from the *Type* drop-down.
18. In the *Options* section, turn on *Override default name* and enter "Downstream_1D" in the field to the right.
19. In the *Events* section, select "100 year" and click **Curve undefined** to bring up the *XY Series Editor* dialog.

20. Outside of SMS, return to the "bc.xls" file in the spreadsheet program.
21. Copy the values from the *Time* column in the "bc.xls" spreadsheet and paste them into *Time (hrs)* column in the *XY Series Editor* dialog.
22. Copy the values from the *Head (m)* column in the "bc.xls" spreadsheet and paste them into *Wse (m)* column in the *XY Series Editor* dialog.
23. Click **OK** to exit the *XY Series Editor* dialog.
24. Click **OK** to exit the *Boundary Conditions* dialog.
25. Turn on " Channel Boundary", " 1D_2D_Connection", and " Cross Sections".

7 Creating Water Level Line Coverage for Output

TUFLOW can generate output that looks like 2D output from the 1D solution. This becomes part of the output mesh and can be viewed inside of SMS. The mesh node locations in this output are determined by water level lines.

To specify the spacing of nodes along the water level lines, do the following:

1. Right-click on " 1D Network" and select **Create Water Level Arcs** to bring up the *Create Water Level Arcs* dialog.
2. In the *Extract options* section, enter "60.0" as the *Distance between WL*.
3. Enter "350.0" as the *WL arc length*.
4. Enter "10.0" as the *Default point distance*.
5. Turn on *Create New Coverage*.
6. Click **OK** to create a " TUFLOW Water Level Lines" coverage and close the *Create Water Level Arcs* dialog.
7. Right-click on " TUFLOW Water Level Lines" and select **Rename**.
8. Enter "Water Level Lines" and press *Enter* to set the new name.
9. Right-click on " Water Level Lines" and select **Trim to code polygon** to bring up the *Select Coverage* dialog.
10. Select " Channel Boundary" from the tree list and click **OK** to close the *Select Coverage* dialog.

All of the water level lines should now be trimmed to the channel boundaries.

11. Select " Water Level Lines" to make it active.
12. Using the **Select Feature Arc** tool, select all the water level lines before the first cross section and press Delete.
13. Click Yes when asked to confirm the deletion.
14. Repeat steps 12–13 for the water level lines after the last cross section.
15. (Figure 14, showing once the lines have been removed).

Water level lines that cross each other create inverted elements. Remove or modify any that are crossing by doing the one of the following:

16. Using the **Select Feature Arc** tool, delete any water level lines that are crossing, or
17. Using the **Select Feature Point** tool, move the endpoints of any water level lines that cross so they don't overlap.
18. Repeat steps 9–10 to trim the waterlines to the channel boundaries.
19. **Frame** the project.

Once all the unneeded or overlapping water level lines have been removed or adjusted, the project should appear similar to Figure 14.

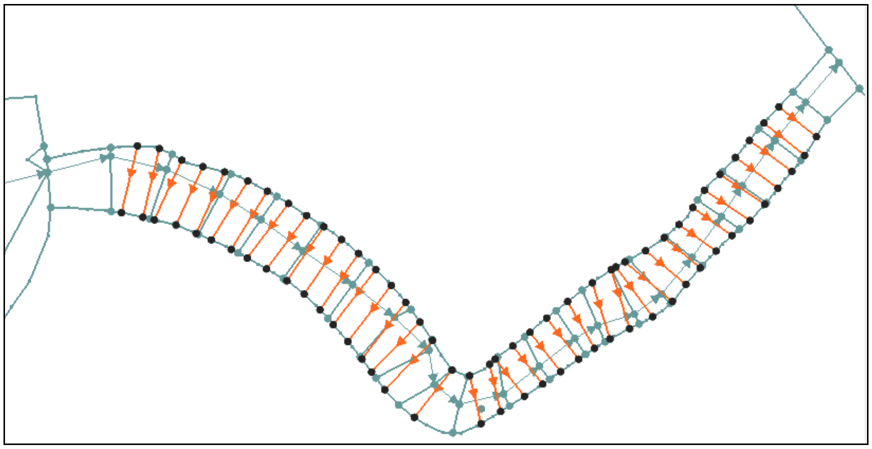


Figure 14 Water level lines

8 TUFLOW Simulation

SMS allows for the creation of multiple simulations, with each including links to these items. The use of links allows these items to be shared between multiple simulations. A simulation also stores the model parameters used by TUFLOW.

To create the TUFLOW simulation:

1. Right-click in an empty part of the Project Explorer and select *New Simulation* | **TUFLOW**.

This creates several new folders and items that will be discussed later.

2. Right-click on " Sim" and select **Rename**.
3. Enter "100year_20m" and press *Enter* to set the new name.

8.1 Geometry Components

Grids are shared through geometry components—as explained in the "TUFLOW 2D" tutorial—by creating and setting them up as follows:

1. Right-click on " Components" and select **New 2D Geometry Component** to create a new " 2D Geom 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 “ 20m”, “ materials”, “ 1d_bc”, and “ Channel Boundary” under “ 20m_geo”.

8.2 Material Definitions

It is necessary to define the material properties by changing the material definitions sets (or values) in the material definitions folder.

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. Using the table below, select each material from the list on the left and enter the value from the table in the *n* field on the right. This is the Manning’s *n* value.
4. 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

8.3 Simulation Setup and Model Parameters

It's necessary to add the items which will be used in the simulation. These items include the geometry component and coverages. Coverages already in the geometry component do not need to be added to the simulation.

Note that a grid must be part of the geometry component or the simulation will not run.

1. Drag the following items, in the listed order, underneath “ 100year_20m”:
   - 20m_geo
   - Cross Sections
   - 1D Network
   - Water Level Lines
   - 1D_2D_Connection

The TUFLOW model parameters include timing controls, output controls, and various model parameters are set up by doing the following:

2. Right-click on “ 100year_20m” and select **2D Model Control…** to bring 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 only *Depth*, *Water Level*, *Flow Vectors*, and *Velocity Vectors*. Turn off all other datasets.
8. In the *Screen/log output* section, turn off *Show water level for a point*.
9. Enter “6” as the *Display interval*.

TUFLOW will write status information every six time steps.

10. Select “Time” from the list on the left.
11. Enter “2” as the *Start Time (hrs)*.
12. Enter “16” as the *Time (hrs)*.
13. Enter “5.0” as the *Time step (s)*.
14. Select “Water Level” from the list on the left.
15. Enter “265.5” as the *Initial Water Level (m)*.
16. Turn on *Override default instability level* and enter “285.0” as the *Instability level (m)*.
17. Select “BC” from the list on the left.
18. Select “100 year” from the *BC event name* drop-down.
19. Click **OK** to close the *TUFLOW 2D Model Control* dialog.

In addition to the 2D model parameters, it's necessary to specify parameters specifically for the 1D portion of the model.

1. Right-click on “ 100year_20m” and select **1D Control** to bring up the *Control 1D* dialog.
2. On the *General* tab, enter “900.0” as the *Output interval (s)*.
3. Enter “265.5” as the *Initial Water Level (m)*.
4. In the *Network* tab, enter “5.0” as the *Depth limit factor*.

This allows water in the channels to be up to five times deeper than the depth of the channel before halting due to a detected instability.

5. Click **OK** to close the *Control 1D* dialog.

9 Saving a Project File

To save all this data for use in a later session:

1. Select *File* | **Save As...** to open the *Save As* dialog.
2. Select “Project Files (*.sms)” from the *Save as type* drop-down.
3. Enter “Cimmaron1d.sms” as the *File Name*.
4. Click **Save** to save project file and close the *Save As* dialog.

10 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 Users Manual*.

2. Right-click on " 100year_20m" and select **Launch TUFLOW** to bring up a console window and launch TUFLOW. This process may take several minutes to complete, depending on the speed of the computer being used.
3. Click **OK** when the prompt states the model run has finished.

11 Using Log and Check Files

TUFLOW generates several files that can be useful for locating problems in a model. In the *data files\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 using the *File* | **View Data File…** command in SMS.

In addition to the text log file, TUFLOW generates paired files in MIF/MID format. These files can be opened in the GIS module of SMS. In the *data files\TUFLOW\runs\log* directory are the files "100year_20m_messages.mif" and "100year_20m_messages.mid". The MIF can be opened in SMS. This file contains messages which are tied to the locations where they occur. Use the **Get Attributes** tool to see the messages at a specific location if they are difficult to read. To use the info tool, simply click on the object while using the tool and the message text or other information is displayed.

The *data files\TUFLOW\check* directory contains several more check files that can be used to confirm that the data in TUFLOW is correct. The **Get Attributes** tool can be used with points, lines, and polygons to check TUFLOW input values.

One of the check files can be used to examine the 1D/2D hydraulic connections. This is the check file ending "1d_to_2d_check.mif". This file includes a polygon for each cell that is along the 1D/2D interfaces (HX arcs). Each polygon (cell) includes data that used by TUFLOW for computing flows between the 1D and 2D domains.

To look at this information:

1. Click **Open** to bring up the *Open* dialog.
2. Browse to the *data files\TUFLOW\checks* directory and select "100year_20m_1d_to_2d_check.mif".
3. Click **Open** to exit the *Open* dialog and bring up the *Mif/Mid import* dialog.
4. In the *Read As* section, select "GIS layer" from the drop-down and click **OK** to close the *Mif/Mid import* dialog and finish importing the MIF file.
5. Right-click in a blank spot in the Project Explorer and select **Uncheck all**.

6. Turn on " 100year_20m_1d_to_2d_check.mif" to make the MIF data visible.
7. Using the **Get Attributes** tool, click on one of the cells in the layer to bring up the *Info* dialog (Figure 15).

The *Info* dialog displays data about the cell, including the bed elevations applicable for the 2D and 1D domains at the cell. The elevation of the 1D bed is interpolated from the node upstream and downstream of the cell location. The 1D nodes on each side and weights used are shown in the dialog under *Primary_Node*, *Weight_to_P_Node*, *Secondary_Node*, and *Weight_to_S_Node*.

8. Select another tool to exit the *Info* dialog.

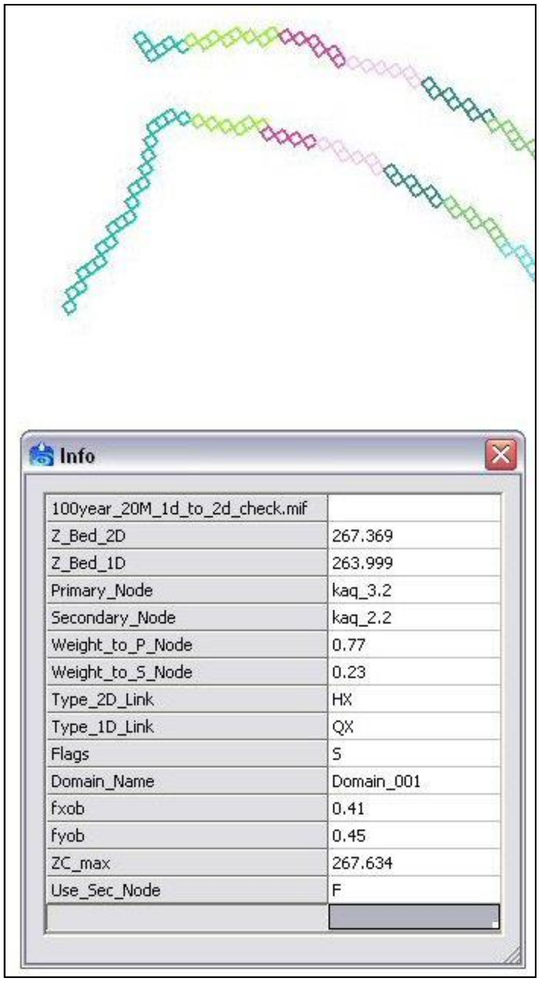


Figure 15 Sample check file

12 Viewing the Solution

TUFLOW has several kinds of output. All the output data is found in the *data files\TUFLOW\results* folder. 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.

In addition to the 1D solution files, the results folder contains a 2DM, MAT, SUP, and several DAT files. These are SMS files which contain a 2D mesh and accompanying solutions. Since water level lines were used, the mesh will also contain solutions for the 1D portions of the model.

To view the solution files from with 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 SUP file and exit the *Open* dialog.
4. If prompted, tell SMS not to overwrite materials with the incoming data.

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

5. Turn off "GIS Data" in the Project Explorer.
6. Turn on and select " Mesh Data" to make it active.
7. Click **Display Options** to open the *Display Options* dialog.
8. Select "2D Mesh" from the list on the left.
9. On the *2D Mesh* tab, turn on *Elements*, *Contours*, and *Vectors*.
10. On the *Contours* tab, in the *Contour method* section, select "Color Fill" from the first drop-down.
11. Click **OK** to close the *Display Options* dialog.

The mesh will be contoured according to the selected dataset and time step. In Figure 16, the square elements represent the 2D portions of the TUFLOW model and the triangular elements represent the 1D portions of the model.

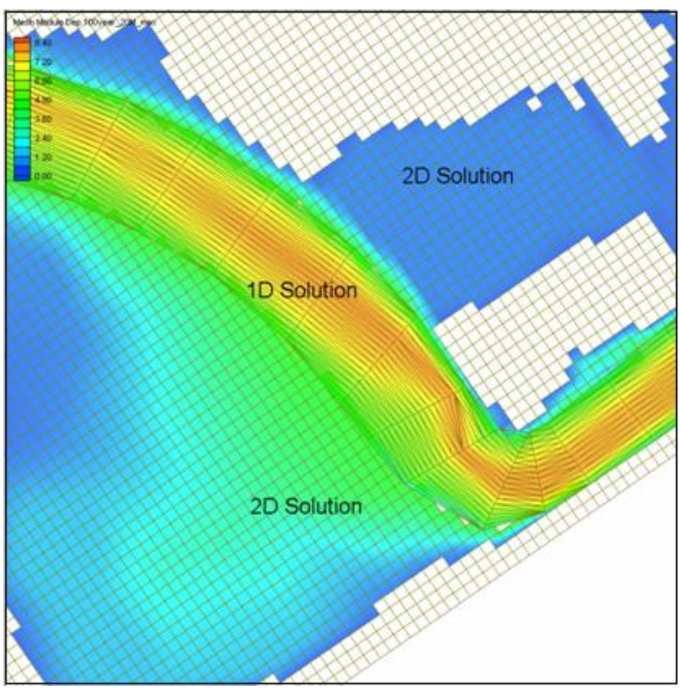


Figure 16 2D and 1D TUFLOW Solution

13 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 a coarse grid would have represented it well. It is possible to force in the higher elevations using a "2D z-lines (advanced)*"* coverage. TUFLOW will use the same grid input files but modify the grid based upon these modifications. The bridge and relief openings are not going to be modeled here. This tutorial assumes the water never reaches the tops of these structures.

For z-lines, 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 "z-polygon", or "2D Z-Lines/polygons coverage", can be used to raise/lower whole regions of cells. The elevation used for a polygon can be set by double-clicking on the polygon using the **Select Feature Polygon** tool.

Define the roadway arcs by doing the following:

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* | **2D Z-Lines (advanced)**.
3. Enter "roadway" as the *Coverage Name*.
4. Click **OK** to close the *New Coverage* dialog.
5. Turn on " ge_hires.jpg" in the Project Explorer.
6. Using the **Create Feature Arc** tool, click out two arcs for the road embankments, as shown in Figure 17.
7. Using the **Select Feature Point** tool, select the top node and enter "274.5" in the *Z* edit field at the top of the SMS window.
8. Select node at the other end of the top arc and enter "274.0" in the *Z* edit field.
9. Select the northern node of the lower road arc and enter "273.5" in the *Z* edit field.
10. Select the southern node of the lower road arc and enter "273.0" in the *Z* edit field.
11. Using the **Select Feature Arc** tool while holding down *Shift*, select both road arcs, then right-click and select **Attributes** to bring up the *Z Shape* dialog.
12. Turn on *Modify Z values*.
13. In the first section, select "Width" from the *Thickness* drop-down and enter "10.0" for *m*.

This changes all elevations within five meters on either side of the z-line.

14. Select "Max" from the *Option* drop-down.

This changes the elevations of cells only if the elevation from the z-line is higher than the original elevation.

15. Click **OK** to close the *Z Shape* dialog.

16. Click **OK** if asked to confirm changing the “Override Z Values”, “Thickness”, and “Option” for each of the two selected arcs.

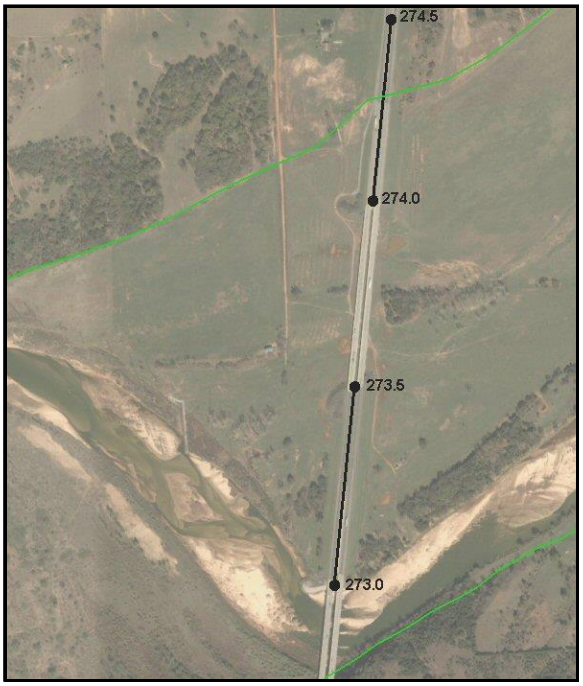


Figure 17 Roadway embankment arc and elevations

14 New Geometry Component and Simulation

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

To create this component:

1. Right-click on “20m_geo” and select **Duplicate** to create “20m_geo (2)”.
2. Right-click on “20m_geo (2)” and select **Rename**.
3. Enter “20m_road” and press *Enter* to set the new name.
4. Drag “roadway” onto the new “20m_road” component.

Similarly, create a new simulation which uses this geometry component by doing the following:

1. Right-click on “100year_20m” and select **Duplicate.**
2. Right-click on “100year_20m (2)” and select **Rename**.
3. Enter “100year_20m_road” and press *Enter* to set the new name.
4. Right-click on “20m_geo” under “100year_20m_road” and select **Delete**.
5. Click **Yes** when asked to confirm deletion.

This deletes the link to " 20m_geo", not " 20m_geo" itself.

6. Drag " 20m_road" onto " 100year_20m_road".

The new simulation will have the same model control and 1D control parameters used previously.

15 Run the New Simulation

Repeating the steps in sections 9 through 12, save the project as "Cimmaron1D_road.sms", export the TUFLOW files, launch TUFLOW, and visualize the results.

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 "Cimmaron1D_road.sms" as the *File Name.*
4. Click **Save** to save the project under the new name and close the *Save As* dialog.
5. Right-click on " 100year_20m_road" and select **Export TUFLOW files**.
6. Right-click on " 100year_20m_road" and select **Launch TUFLOW** to bring up a console window and launch TUFLOW. This process may take several minutes, depending on the speed of the computer being used.
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 import the SUP file, exit the *Open* dialog, and bring up the *Select Tree Item for Datasets* dialog.

This is 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* dialog.
12. Review the " 100year_20m_road" solution datasets and click through the time steps to see the results.

16 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.

This concludes the "TUFLOW 1D/2D" tutorial. Feel free to continue experimenting with the SMS interface, or exit the program.

