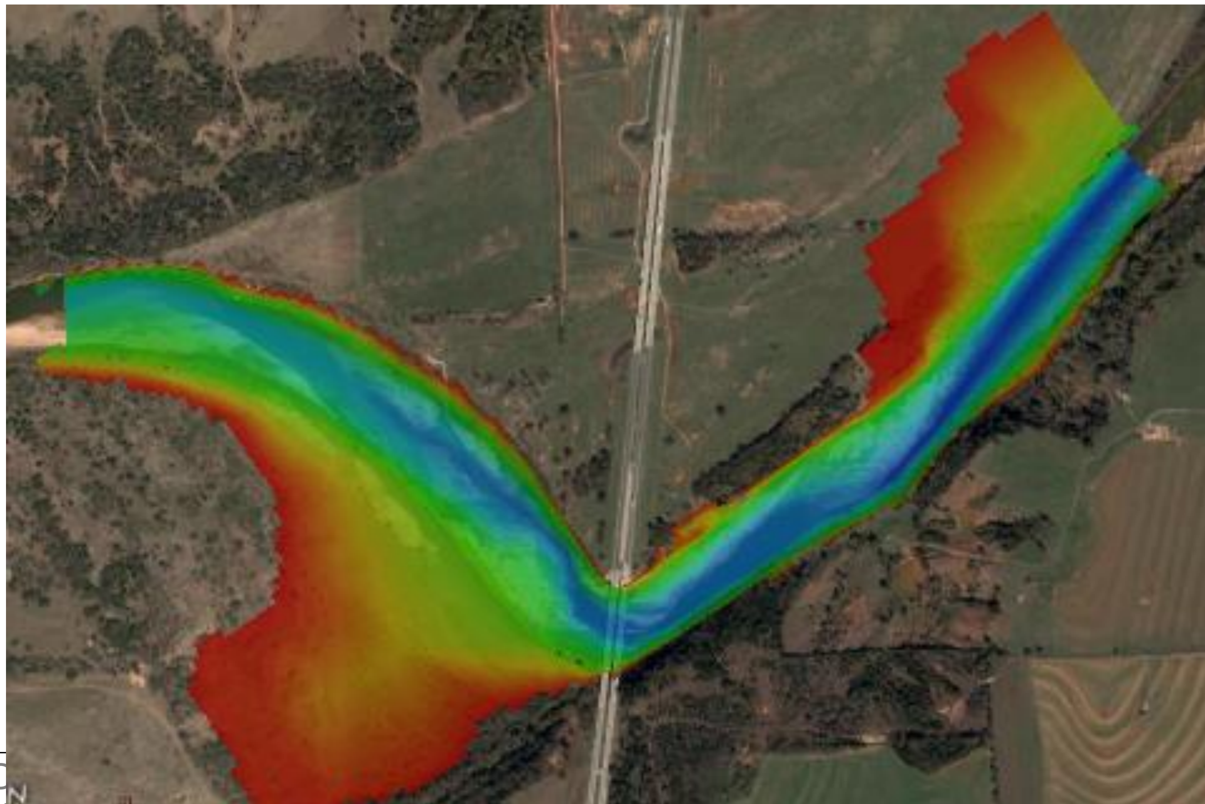
v. 12.2

SMS 12.2 Tutorial

TUFLOW 1D/2D



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

1	**Introduction and Background Data**	**2**
2	**1D/2D TUFLOW models**	**3**
3	**Defining the 2D Portion of the Model**	**5**
3.1	2D Computation Domain	5
4	**Setting up the 1D Network**	**6**
4.1	Creating Cross Sections	8
5	**Defining the 1D/2D Connection**	**13**
5.1	1D/2D Flow Interfaces	14
5.2	1D/2D Connections	14
6	**Specifying the boundary conditions**	**18**
6.1	2D Downstream Water Level Boundary Condition	18
6.2	Creating The 1D BC	20
7	**Creating Water Level Line Coverage for Output**	**21**
8	**TUFLOW Simulation**	**22**
8.1	Geometry Components	23
8.2	Material Definitions	23
8.3	Simulation Setup and Model Parameters	23
9	**Saving a Project File**	**25**
10	**Running TUFLOW**	**25**
11	**Using Log and Check Files**	**25**
12	**Viewing the Solution**	**27**
13	**Including the Roadway in the Model**	**29**
14	**New Geometry Component and Simulation**	**31**
15	**Run the New Simulation**	**31**
16	**Conclusion**	**32**

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: http://www.tuflow.com.

This tutorial focuses on adding 1D cross sections to a 2D model of where the Cimarron River crosses I-35 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 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 the file "Cimmaron_1D.sms" in the *data files* folder for this tutorial and click **Open**. This will load an SMS project with a background image, elevation data, the 20m grid created in the 2D tutorial, and three map coverages, as seen in Figure 1 below.

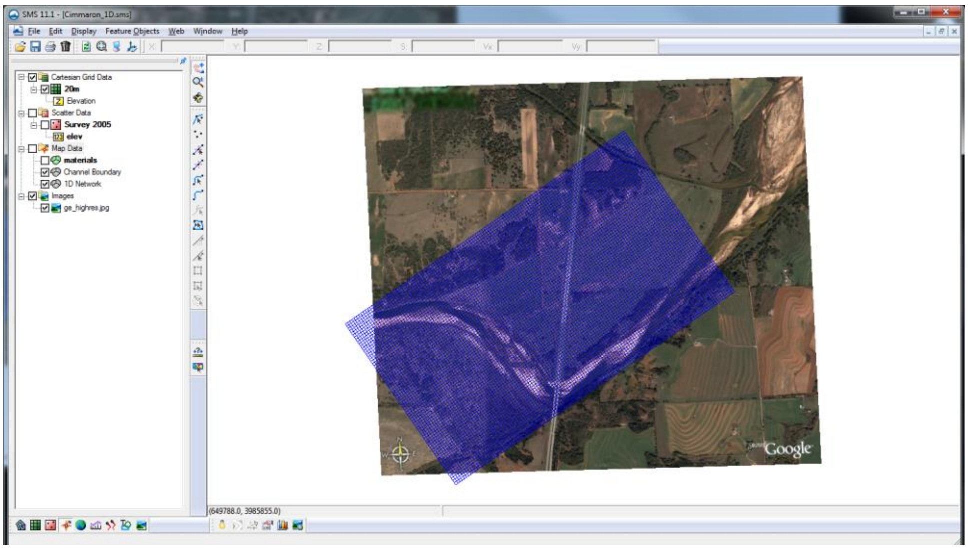


Figure 1 View of Map and Image Data

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

This tutorial illustrates the third of these methods.

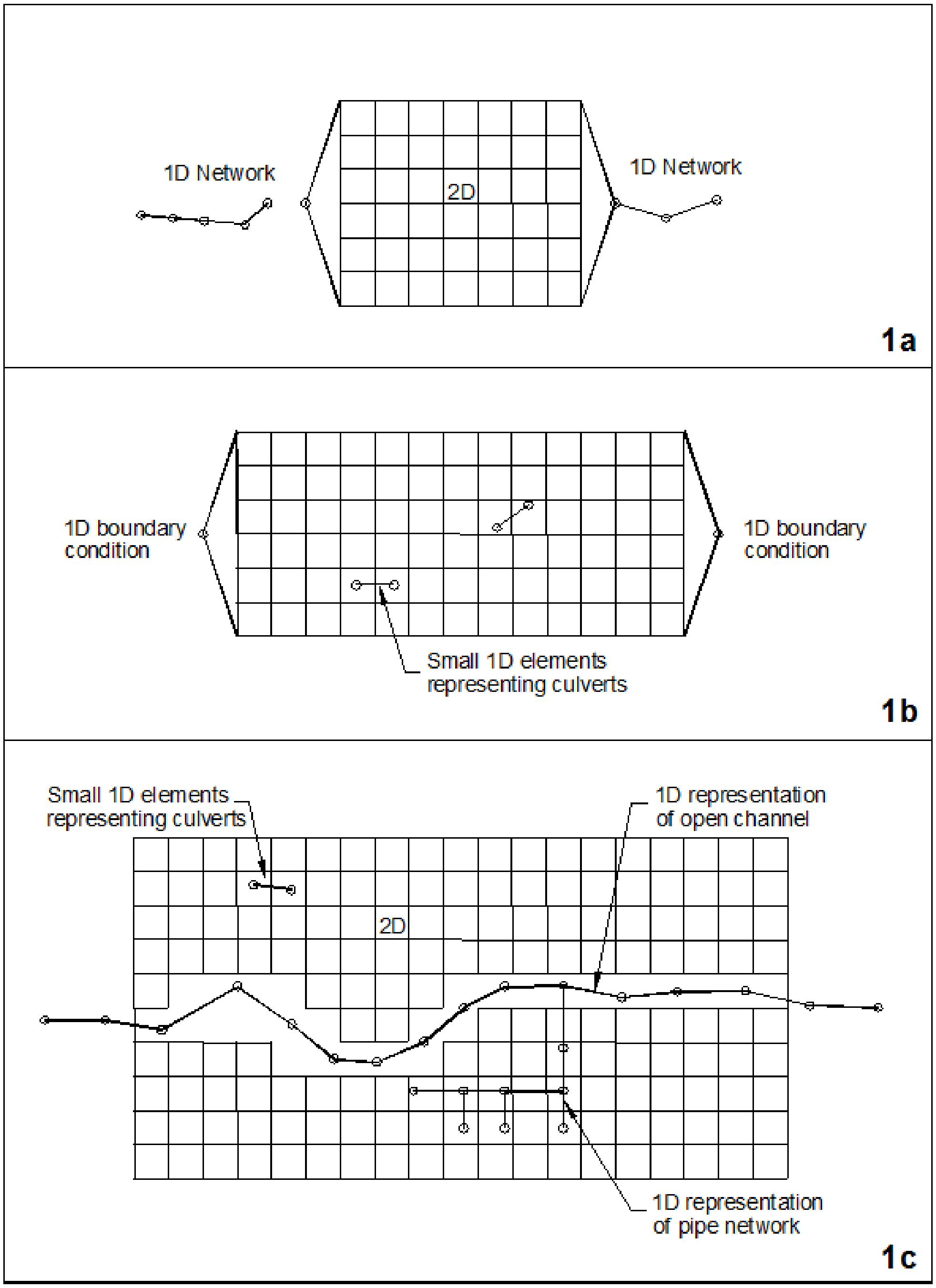


Figure 2 1D/2D linking mechanisms (from TUFLOW Users Manual, 2010 (Build 2010-10-AB), p.3–4 "The Modelling Process", www.tuflow.com)

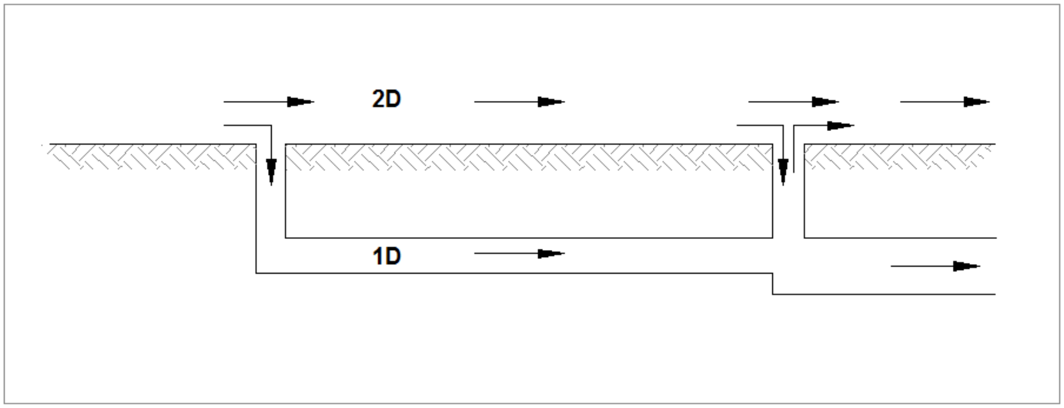


Figure 3 Modelling a pipe system in 1D underneath a 2D domain (from TUFLOW Users Manual, 2010 (Build 2010-10-AB), p.3–5 "The Modelling Process", www.tuflow.com)

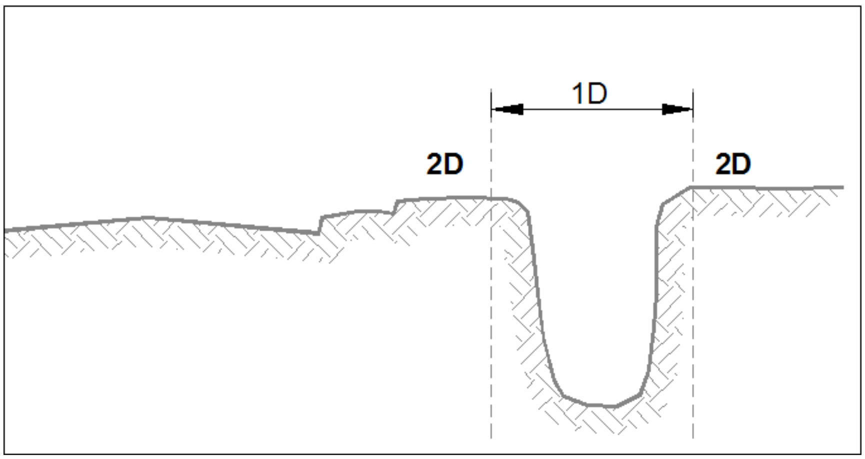


Figure 4 Modelling a channel in 1D and the floodplain in 2D (from TUFLOW Users Manual, 2010 (Build 2010-10-AB), p.3–5 "The Modelling Process", www.tuflow.com)

3 Defining the 2D Portion of the Model

The TUFLOW 2D tutorial demonstrated how to define a grid to represent the geographical features in a study domain such as a flood plain. 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 the *Channel Boundary* coverage.

1. Click on the "Channel Boundary" coverage to display this polygon.

Note that the orange polygon encloses the channel and the regions both upstream and down stream 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 the “Channel Boundary” coverage and select *Type* | *Models* | *TUFLOW* | **1D/2D BCs and Links**.
3. Click the **Select Feature Polygon** tool and double-click inside the channel to bring up the *Boundary Conditions* dialog.
4. Set the *Type* to “No BC” and turn on the *Set cell code* option under *Options.*
5. Select “Inactive -- not in mesh” from the *Set cell code* 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. Uncheck the box next to the "20m" grid in the Project Explorer to reduce the amount of data visible on the screen.
2. Click on the coverage named “1D Network” to make it active.
3. Right-click on the “1D Network” coverage and select *Type* | *Models* | *TUFLOW* | **1D Networks**.
4. Using the **Create Feature Arc** tool, create a series of arcs (one for each node pair). This means that each arc will start and stop at each node. Create the nodes working from left to right.

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 look something like Figure 5.

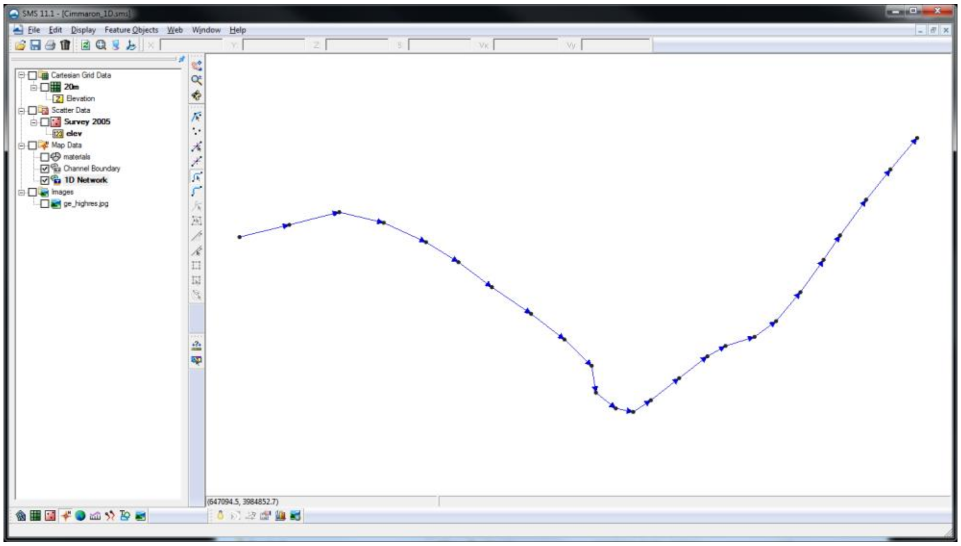


Figure 5 Creating the 1D Network centerline

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.

1. Using the **Select Feature Arc** tool select the first arc in the network.
2. Right-click and select **Attributes** to bring up the *Channel Attributes* dialog.

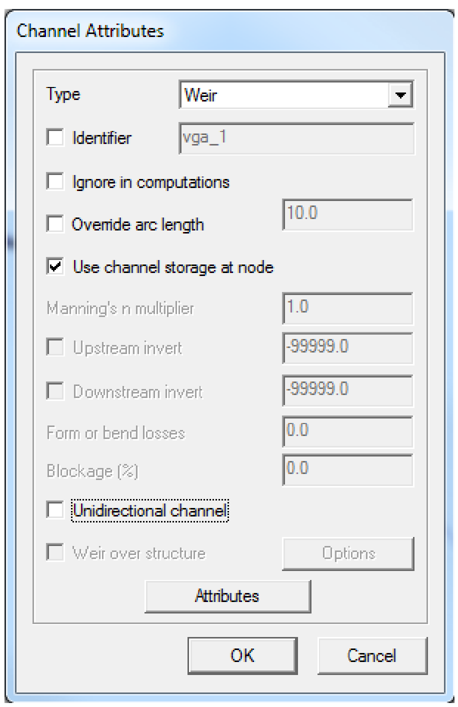


Figure 6 Channel Attributes dialog

3. Change the *Type* to “Weir” The dialog should look like Figure 6.
4. Click on **Attributes** button at the bottom of the dialog to bring up the *Weir Attributes* dialog.
5. Select the Define rectangular section radio button.
6. Set *Invert?* in the *Geometry* section to “264.0” (the elevation of the channel).
7. Set *Width* to “1000” (wide enough to cover the majority of the floodplain).
8. Click **OK** to exit the *Weir Attributes* dialog.
9. Click **OK** to exit the *Channel Attributes* dialog.

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 or 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. For the *Coverage Type*, select “1D Cross Sections” under the TUFLOW folder.
3. Enter “Cross Sections” in *Coverage Name*.
4. Click **OK** to bring up the *CsDb Management* dialog.
5. Click **OK** to close the *CsDb Management* dialog. The cross section data will be added later.
6. Go to *Display* | **Display Options** to bring up the *Display Options* dialog.

7. Select "Map" from the list on the left and turn on *Inactive coverage*.
8. Click **OK** to close the *Display Options* dialog.
9. In the Project Explorer, turn off the following so only the feature 1D network data is visible (some may already be off):
   - "materials" coverage
   - "Cartesian Grid Data"
   - "Survey 2005" scatter dataset
   - "ge_highres.jpg" GIS Data
10. Right-click the "1D Network" coverage and choose the **Create Cross Section Arcs** option. The *Create Cross Section Arcs* dialog will appear.
11. Turn off the *Midpoints* toggle.
12. Change *Cross section lengths* to "350". This will ensure 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.

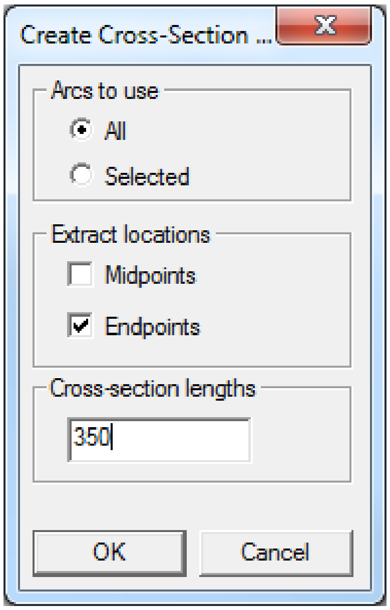


Figure 7 Values for creating cross section arcs from 1D Network

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. Go through the cross sections and adjust arcs as seems fit by using the **Select Feature Point**  tool and selecting nodes. Move the nodes so the cross sections do not overlap. The worst problems are generally at the bend. 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. The very first channel is the weir and so the cross section at the first node is not needed. Delete it by selecting it with the **Select Feature Arc**  tool and pressing the *Delete* key.
3. A dialog will appear asking to confirm the deletion. Click **Yes**.
4. The first and the last cross sections should connect with the extents of the 2D domain. Drag the endpoints of these arcs to the nodes in the "Channel Boundary" coverage.

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 “Cross Sections” and choose **Trim to Code Polygon**.

This will trim 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. If working with a project that uses more than one such coverage, a dialog will appear to allow choosing a coverage.

When done, the cross sections should look like Figure 8.

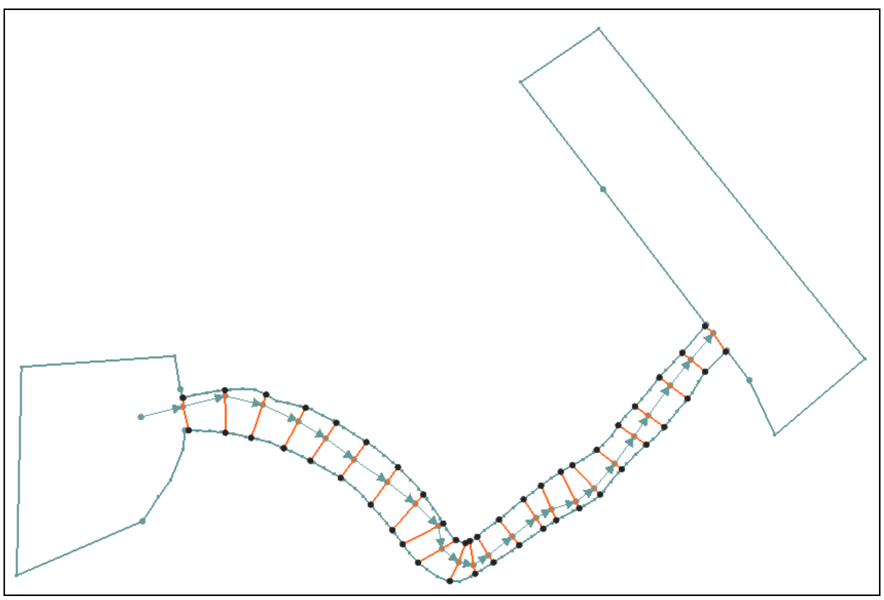


Figure 8 Final trimmed cross section arcs

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

1. Right-click on the "Cross Sections" coverage and select **Extract from Scatter**. This will extract elevation data from the active dataset from the active scatter set (TIN), though there will be no visible changes on the screen.
2. Right-click on "Cross Sections" again and select **Map Materials From Area Coverage** to bring up the *Select Coverage* dialog.
3. Select "materials".
4. Set the *Default material* to "channel".
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:

1. Click the **Select Feature Arc**  tool and double-click on one of the cross section arcs. A *TUFLOW Cross Section* dialog appears showing the cross section ID.
2. Click on the **Edit** button 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 9 is an example of the dialog.

In 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.

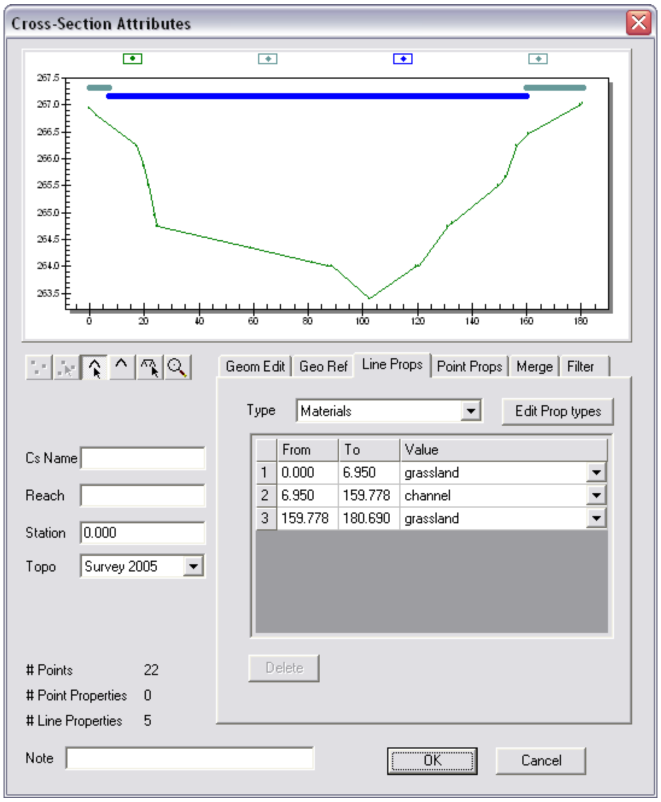


Figure 9 Cross Section Attributes dialog

Clicking on the *Line Props* tab shows the materials that are assigned to each section of the cross section. The material breaks may be edited in this dialog using the tools in the plot window or the spreadsheet below. If there is additional information that is wanted to be incorporated into the cross section, that would be done 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.

3. Since there is no additional data, click **Cancel** to close the *Cross Section Attributes* dialog.
4. 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.

1. Select *Display* | **Plot Wizard…** to bring up the *Plot Wizard*.
2. Select "TUFLOW Cross Section" from the list on the left and click **Finish** to close the *Plot Wizard*.

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

3. Select a cross section with the **Select Feature Arc** tool. The cross section profile will appear in "Plot 1".

4. 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 10).

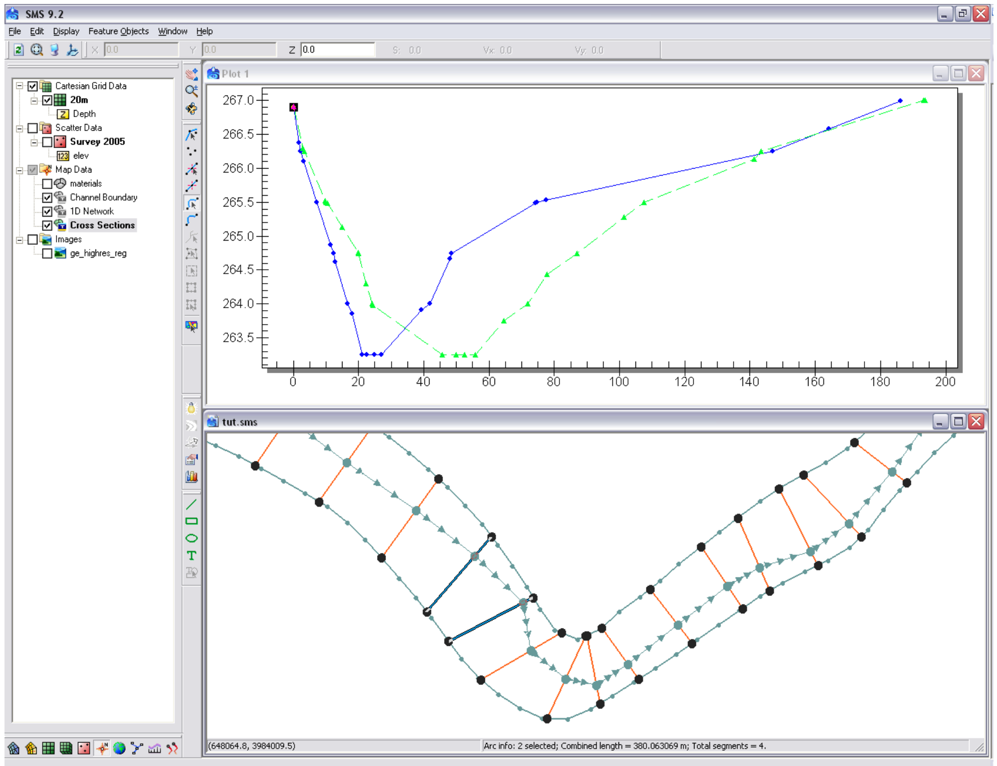


Figure 10 TUFLOW cross section plot

5. 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).

6. 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.

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. Click on the "Channel Boundary" coverage to make it active.
2. Select the **Select Feature Arc** tool and, using the *Shift* key, select the two arcs running along both banks of the channel and the single arc on each side of the upstream (left) end of the model (Figure 11). 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.

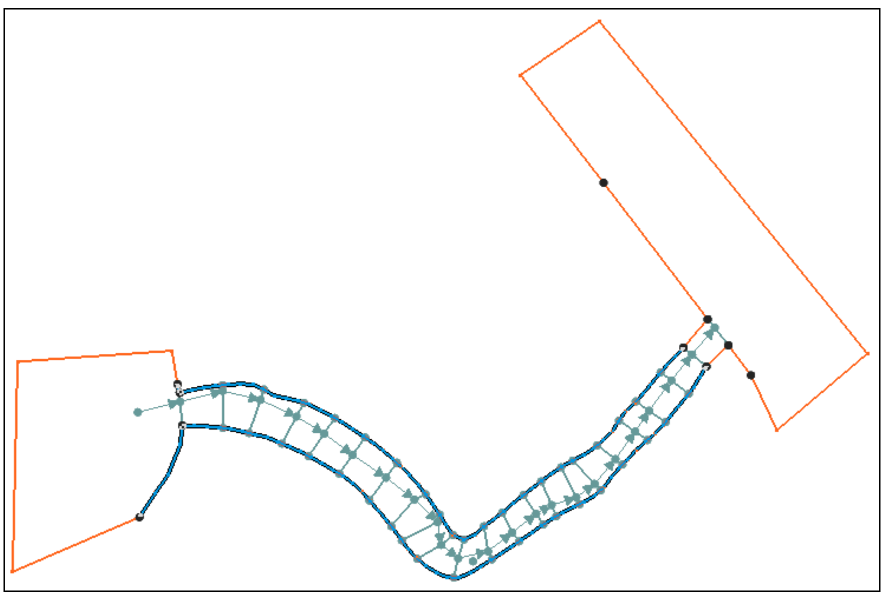


Figure 11 Selected HX arcs

3. Right-click and select **Attributes** to bring up the *Boundary Conditions* dialog.
4. Set *Type* to "1D Flow/2D Water Level Connection (HX)".
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 the “Cross Sections” coverage and select **Duplicate**. This creates a coverage named "Cross Sections (2)".
2. Click on the new coverage to make it active.
3. Right-click on it and **Rename** it to “1D_2D_Connection”.
4. Right-click again and select *Type* | *Models* | *TUFLOW* | **1D–2D Connections**.
5. Using the **Select Feature Vertex** tool, right-click in an area where nothing is and choose **Select All**. This will select all of the vertices. There is one vertex at the center of each cross section.
6. Right-click again and select the **Convert to Nodes** option. This splits each of the arcs into two arcs, creating separate arcs connecting each node on the centerline to the HX arcs.
7. Right-click on the *1D_2D_Connection* Coverage and select **Properties** to bring up the *Select Boundary Condition Coverage* dialog.
8. Select “Channel Boundary” under Map Data in the tree.
9. Click **OK** to close the *Select Boundary Condition Coverage* dialog.

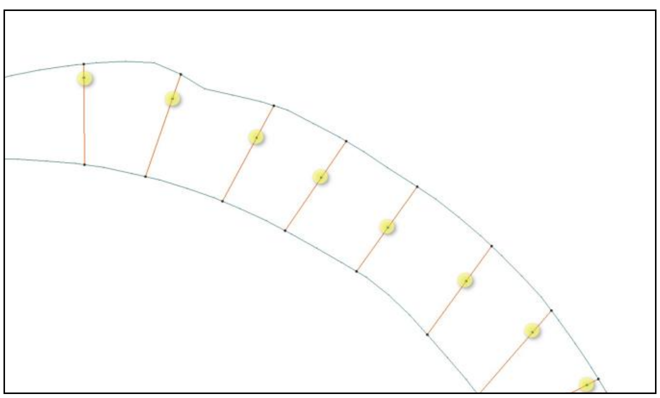


Figure 12 1D/2D connection arcs (highlighted) to be converted into nodes

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

1. Right-click on the “1D_2D_Connection” coverage and select **Clean Connections**. This brings up the *Clean Options* dialog.
2. Enter "5.0" in the *Tolerance* field.

3. Turn on the *2D BC Coverage (HX Lines)* option.
4. Highlight "Channel Boundary" in the Map Data tree.
5. Click **OK** to close the *Clean Options* dialog.

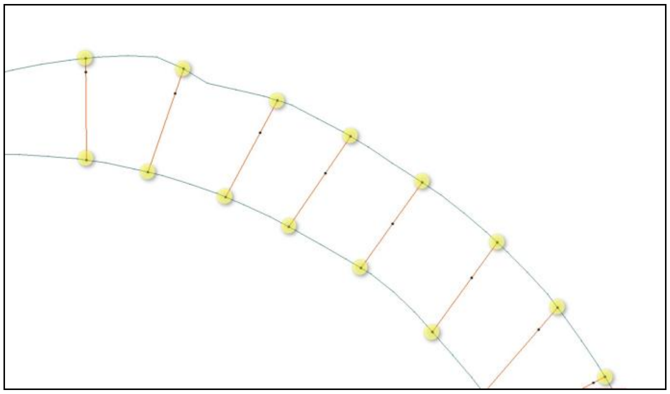


Figure 13 Vertices created or moved (highlighted) by Clean Connections command

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. See Figure 13, above.

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:

1. Select the "1D_2D_Connection" coverage to make it active.
2. Click on the **Create Feature Arc** tool and digitize two arcs connecting the upstream side of the 2D domain to the downstream node of the 1D weir boundary (Figure 14).

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.

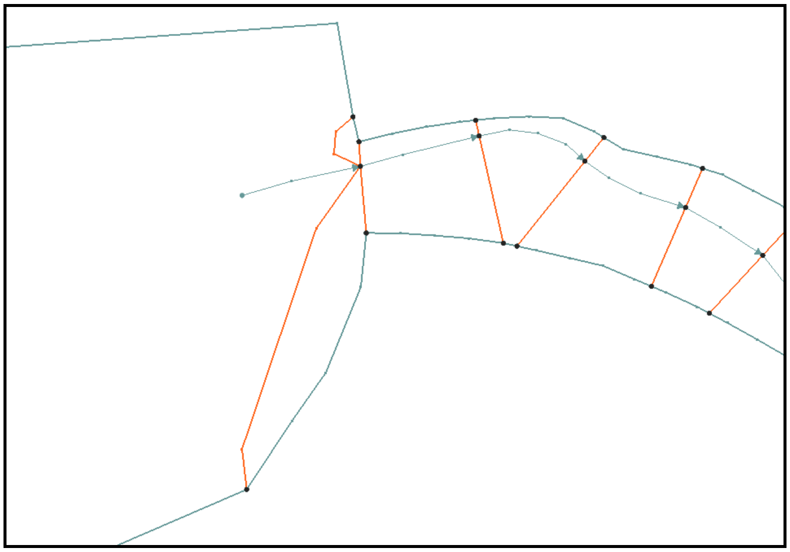


Figure 14 1D/2D Connection Arcs added at upstream end.

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:

1. Click on the **Select Feature Arc** tool.
2. Select then *Delete*, the two most downstream 1D/2D connection arcs (Figure 15).

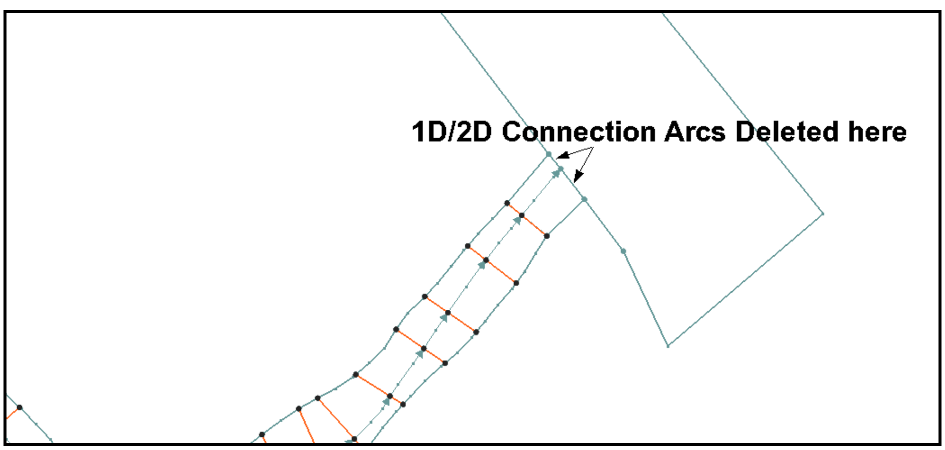


Figure 15 Location of deleted 1D/2D connection arcs on downstream end.

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.

1. Click on the "Channel Boundary" coverage to make it active.
2. Using the **Select Feature Arc** tool, select the most upstream arc (Figure 16).
3. Double-click on the arc to bring up a *Boundary Conditions* dialog.
4. Change *Type* to "1D Flow/2D Water Level Connection (HX)".
5. Click **OK** to close the *Boundary Conditions* dialog.

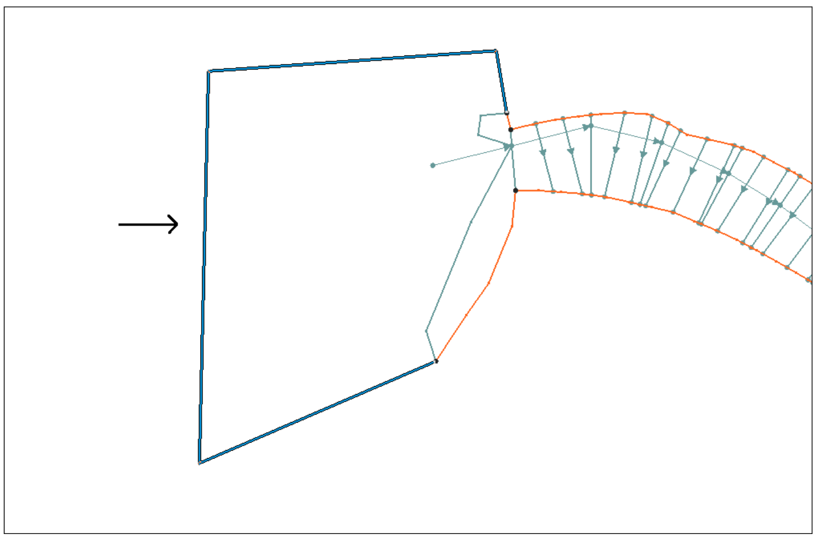


Figure 16 Channel boundary arc to select

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 the "Channel Boundary" coverage to make it active.
2. Using the **Select Feature Arc** tool and the *Shift* key, select the two arcs along the downstream side of the 2D domain as shown in Figure 17.
3. Right-click and choose **Attributes** to bring up a *Boundary Conditions* dialog.
4. Set *Type* to "Wse vs Time (HT)".
5. Click **Edit Events** to bring up the *TUFLOW BC Events* dialog.
6. Click the **Add** button and an event named "new_event" will appear.
7. Double-click on "new_event" and change its name to "100 year".
8. Click **OK** to close the *TUFLOW BC Events* dialog.
9. Select the "100 year" event and click on the large **Curve undefined** button. This brings up the *XY Series Editor*.
10. Open the "BC.xls" file in a spreadsheet editor.
11. Copy the values from the *Time* column in the spreadsheet and paste them into *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 to something more intuitive (Figure 17):

1. Double-click on the arc to the left of the channel to bring up the *Boundary Conditions* dialog.
2. Turn on *Override default name* and enter "downstream_wl_left" in the field to the right.
3. Click **OK**.
4. Repeat steps 1-3 for the arc to the right, entering "downstream_wl_right" as the name.

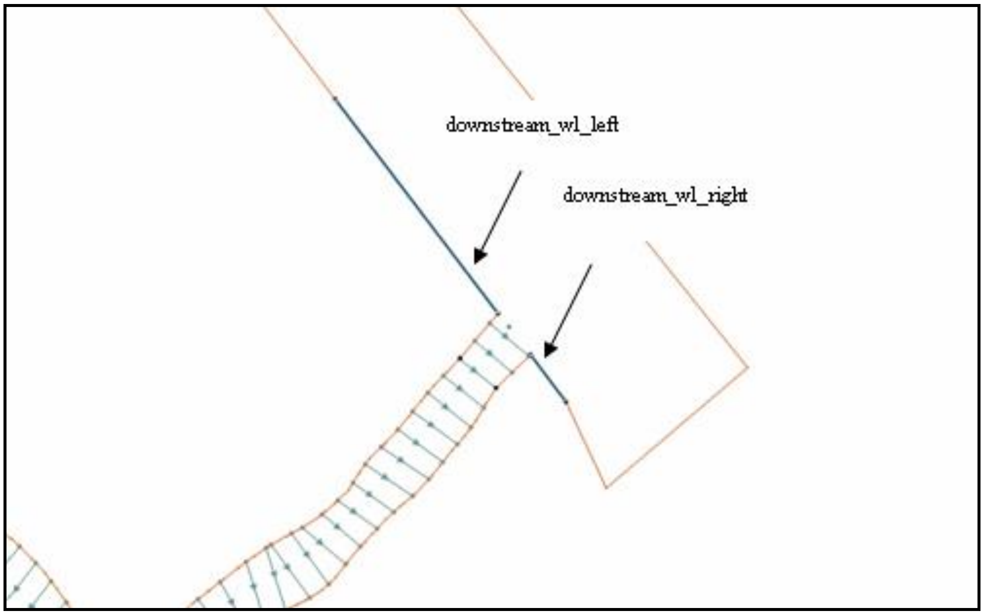


Figure 17 Location of downstream boundary condition arcs

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 Map Data and select **New Coverage** to bring up the *New Coverage* dialog.
2. Under *Coverage Type*, select “1D–2D BC and Links” under the TUFLOW folder.
3. Enter "1d_bc" for the *Coverage Name*.
4. Click **OK** to close *New Coverage* dialog.
5. Select the new “1d_bc” coverage 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. (If the simulation is very crowded, turn off all the coverages in the Project Explorer except “1D network” and “1d_bc” so to not snap to nodes in other coverages.)
7. Using the **Select Feature Point** tool, double-click the node at the upper end of the river to bring up the *Boundary Conditions* dialog.
8. Change the *Type* to “Flow vs Time (QT)” and click the *Override default name* checkbox and type “Upstream_1D”.

9. Choose “100 year” under *Events*.
10. Click on the large **Curve undefined** button to bring up the *XY Series Editor*.
11. 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.
12. 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.
13. Click **OK** to close the *XY Series Editor* dialog.
14. Click **OK** to close the *Boundary Conditions* dialog.
15. Now double-click the downstream node to open the *Boundary Conditions* dialog.
16. In the dialog, make the *Type* “Wse vs Time (HT)” and click the *Override default name* checkbox and type “Downstream_1D”.
17. Choose “100 year” under *Events*.
18. Click on the large **Curve undefined** button to bring up the *XY Series Editor*.
19. 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.
20. 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.
21. Click **OK** to exit the XY Series Editor dialog.
22. Click **OK** to exit the *Boundary Conditions* dialog.

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 the “1D Network” and select **Create Water Level Arcs** to bring up the *Create Water Level Arcs* dialog.
2. Set the *Extract options* as follows:
   - *Distance between WL* to “60.0”.
   - *WL arc length* to “350.0”.
   - *Default point distance* is “10.0”.
3. Turn on Create New Coverage.
4. Click **OK** to close the *Create Water Level Arcs* dialog.

5. Right-click on the new coverage and select the **Rename** command to enter "Water Level Lines" as the coverage name.
6. Right-click the coverage and select **Trim to code polygon**. A *Select Coverage* dialog will appear.
7. Select "Channel Boundary" under the Map Data tree.
8. Click **OK** to close the *Select Coverage* dialog.
9. Use the **Select Feature Arc** tool to *Delete* any water level lines above the first cross section or below the last cross section (Figure 18, showing once the lines have been removed).
10. Crossing water level lines create inverted elements. Delete crossing water level lines (especially at the bend) or drag their endpoints so they don't overlap (Figure 18).

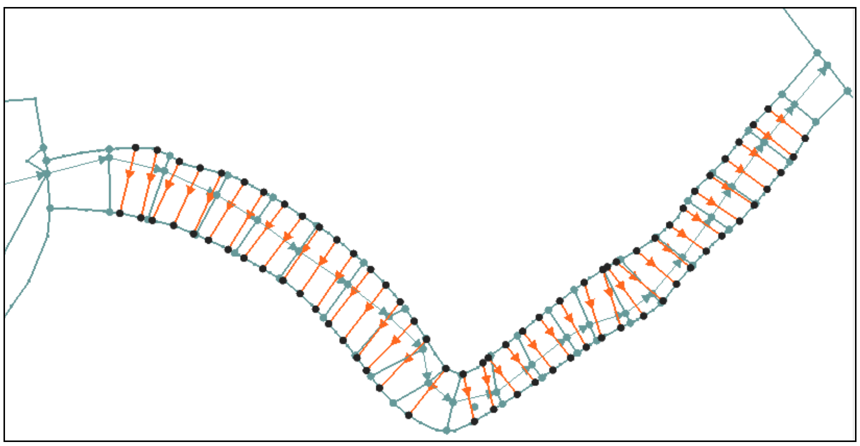


Figure 18 Water level lines

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 the empty part of the Project Explorer and choose *New Simulation* | **TUFLOW**. This will create several new folders that will be discussed later. 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 **Rename** then enter "100year_20m".

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 the folder named “Components” and choose **New 2D Geometry Component**.
2. Right-click on the new “2D Geom Component” item and select **Rename** then enter “20m_geo” as the new name.
3. Drag the “20m” grid and the “materials”, “1d_bc”, and “Channel Boundary” coverages under the “20m_geo” component tree item.

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 the “Material Sets” folder and select **New Material Set** from the menu. A new “Material Set” will appear.
2. Right-click on the new “Material Set” and select **Properties** from the menu to bring up the *TUFLOW Material Properties* dialog.
3. Change the values for Manning’s *n* for all the materials in the list on the left of the dialog according to the table below.
4. Click **OK** to close the *TUFLOW Material Properties* dialog when finished.

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.

1. Drag the following items underneath the “100year_20m” simulation in the Project Explorer:
   - The geometry component (“20m_geo”).
   - The following coverages: “Cross Sections”, “1D Network”, “Water Level Lines”, and “1D_2D Connection”.

Note that a “20m” grid must be part of the geometry component or the simulation will not run.

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

2. Right-click on the “100year_20m” simulation 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, 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”.
    - “Flow Vectors” (unit flowrate).
    - “Velocity Vectors”.
6. In the *Screen/Log Output* section:
    - Turn off *Show water level for a point.*
    - Set *Display interval* to “6”. TUFLOW will write status information every six time steps.
7. Select the *Time* tab from the list on the left and set:
    - *Start Tim (hrs)* “2”.
    - *End Time (hrs)* to “16”.
    - *Time step (s)* to “5.0”.
8. Select *Water Level* from the list on the left and do the following:
    - Set *Initial Water Level (m)* to “265.5”.
    - Toggle on *Override Default Instability Level*.
    - Set *Instability level (m)* to “285.0”.
9. Select *BC* from the list on the left.
10. Select “100 year” from the *BC event name* drop-down list.
11. Click **OK** to close the *TUFLOW 2D Model Control* dialog.

In addition to the normal 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, set:
   - *Output interval (s)* to "900".
   - *Initial Water Level (m)* to "265.5".
3. In the *Network* tab, change the *Depth limit factor* to "5.0". 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.
4. 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. Enter a *File Name* of "Cimmaron1d.sms".
3. Click the **Save** button to save the files.

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 the simulation and select **Export TUFLOW files**. This will create a directory named TUFLOW where the files will be written. The directory structure models that described in the *TUFLOW Users Manual*.
2. Right-click on the 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 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 will be found 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. If the messages are difficult to read, use the **Get Attributes** tool to see the messages at a specific location. To use the info tool, simply click on the object 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. Use *File* | **Open** to bring up the *Open* dialog.
2. Select "100year_20m_1d_to_2d_check.mif" from the *data files\TUFLOW\checks* directory and click the **Open** button. If prompted, choose to open the file as a GIS layer.
3. Turn off all other display items by right-clicking at the bottom of the Project Explorer and selecting "uncheck all."
4. Turn on the tree item for the GIS layer just loaded.
5. Using the **Get Attributes** tool, click on one of the cells in the layer.
6. An *Info* dialog will come up displaying data about the cell as in Figure 19. This information includes 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.
7. Select another tool to exit the *Info* dialog.

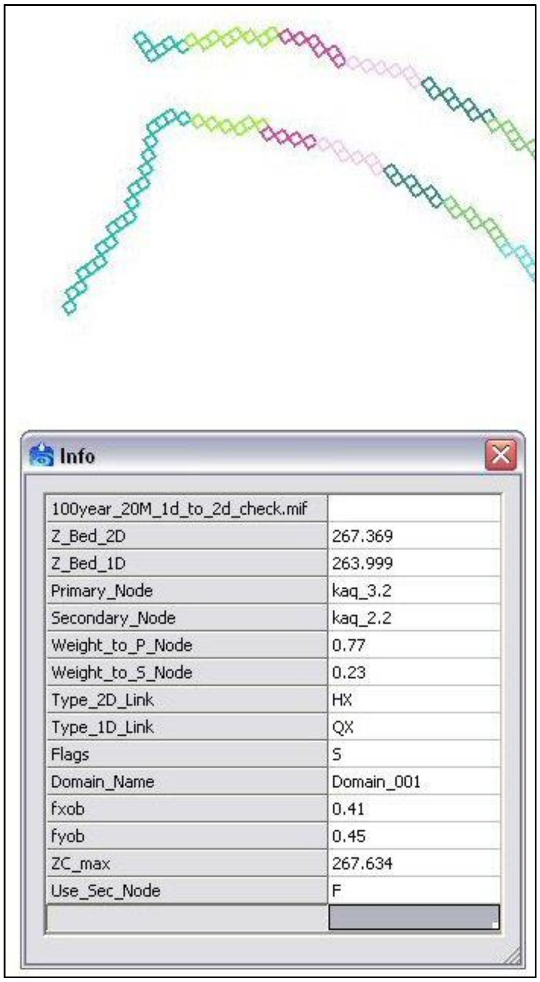


Figure 19 Sample check file

12 Viewing the Solution

TUFLOW has several kinds of output. All the output data is found in *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. Select *File* | **Open** to bring up the *Open* dialog.

2. Browse to the *results* folder from the TUFLOW directory and locate the "100year_20m.xmdf.sup" file and and click **Open**.
3. If prompted, tell SMS not to overwrite materials with the incoming data.

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

4. From the Project Explorer, turn off all Map Data, Scatter Data, and Cartesian Grid Data. Turn on and highlight the Mesh Data.
5. Click the **Display Options** to open the *Display Options* dialog.
6. Select the *2D Mesh* item from the list on the left then turn on *Elements*, *Contours,* and *Vectors*.
7. Switch to the *Contours* options tab and select "Color Fill" as the *Contour Method*.
8. Click **OK** to close the *Display Options* dialog.

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

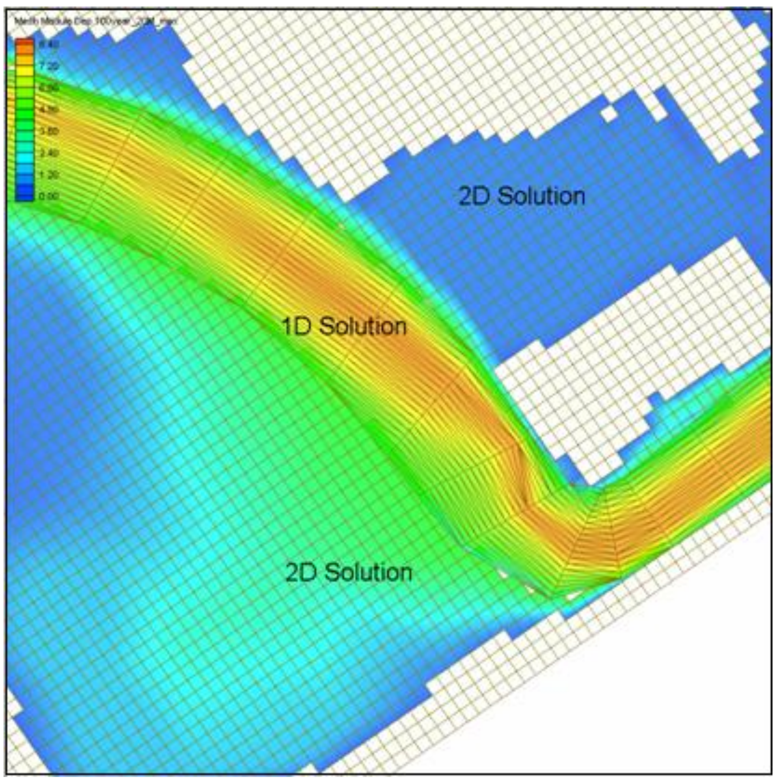


Figure 20 2D and 1D TUFLOW Solution

In Figure 20, the square elements represent the 2D portions of the TUFLOW model and the triangular elements represent the 1D portions of the model.

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, assuming 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 polygon tool.

To define the roadway arc:

1. Right-click Map Data in the Project Explorer and selecting **New Coverage** to bring up the *New Coverage* dialog again.
2. Select "2D Z-Lines (advanced)" under *Coverage Type* and enter "roadway" as the *Coverage Name*.
3. Click **OK** to close the *New Coverage* dialog.
4. Display the "ge_hires.jpg" map image by selecting its box in the Project Explorer.
5. Using the **Create Feature Arc** tool, click out two arcs for the road embankments, as shown in Figure 21.

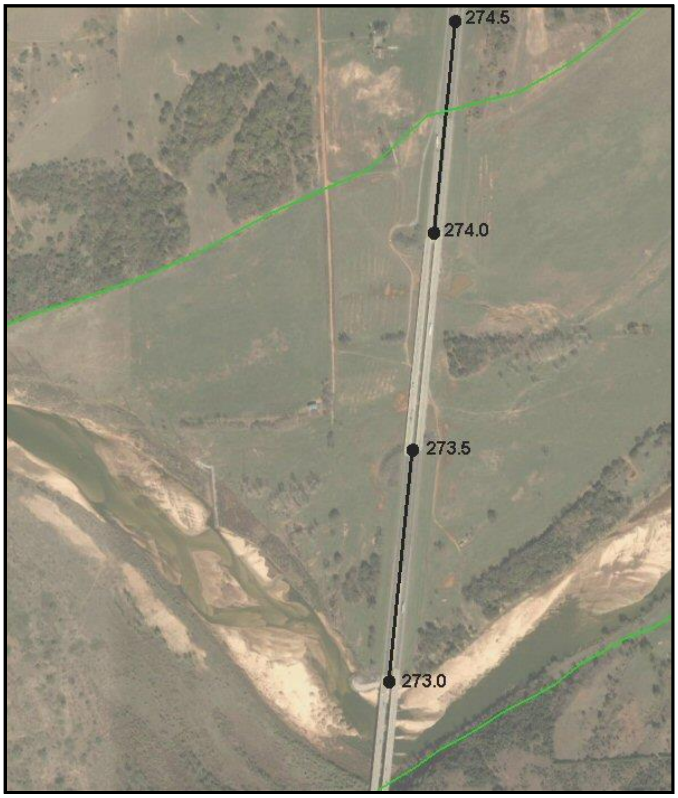


Figure 21 Roadway embankment arc and elevations

6. Using the **Select Feature Point** tool, then select each node in turn and set the elevation for each in the Z edit field, as indicated below (from top to bottom):
   - "274.5".
   - "274.0".
   - "273.5".
   - "273.0".
7. Using the **Select Feature Arc** tool, select both newly created arc by using the *Shift* key, then right-click and choose **Attributes** to bring up the *Z Shape* dialog.
8. Turn on *Modify Z values*.
9. Set *Thickness* to "Width" and change *m* to "10.0". This will change all elevations within 5 meters on either side of the z-line.
10. Set *Option* to "Max". This will make it so it will only change the elevations of cells if the elevation from the z-line is higher than the original elevation.
11. Click **OK** to close the *Z Shape* dialog.
12. Click **OK** if a dialog appears asking if wanting to change the "Override Z Values", "Thickness", and "Option" for each of the two selected arcs.

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 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 is required.

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 new “20m_road” component.

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

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 simulation labeled “20m” and select **Delete**.
4. Click **Yes** when asked if you want to delete it. This deletes the link to the grid component, not the component itself.
5. Drag the new geometry component “20m_road” into the new “100year_20m_road” simulation.

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

15 Run the New Simulation

1. 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.
2. Select *File* | **Save as...** to bring up the *Save as* dialog.
3. Enter a *File Name* of “Cimmaron1D_road.sms” and click **Save**.
4. Right-click on the “100year_20m_road” simulation and select **Export TUFLOW files**.
5. 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.
6. Click **OK** when the dialog indicates the simulation is finished.

7. Select *File* | **Open** to bring up the *Open* dialog again.
8. Browse to the *data files\TUFLOW\results* folder.
9. Select “100year_20m_road.xmdf.sup” file and click **Open**.
10. 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**.
11. 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. Continue to experiment with the SMS interface or may quit the program.