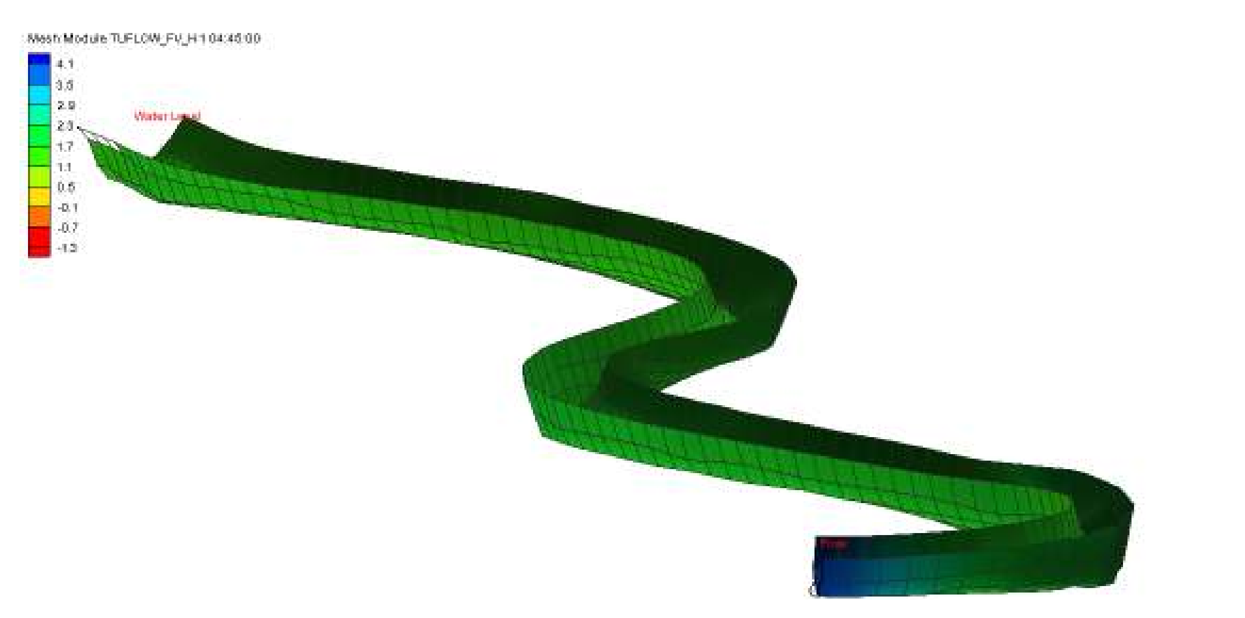
SMS

v. 11.1

SMS 11.1 Tutorial

TUFLOW FV



Objectives

TUFLOW FV is an engine for performing 2D and 3D hydrodynamic simulations. The model solves the Non-linear Shallow Water Equations (NLSWE) on a flexible mesh using a finite-volume numerical scheme.
In this tutorial a simple model of a short section of river is created using the SMS TUFLOW FV interface. We will be building a mesh for an inbank area of a river, and we will apply an upstream inflow boundary and a downstream tidal boundary.

Prerequisites

- None

Requirements

- Map Module
- Mesh Module
- Scatter Module
- TUFLOW FV

Time

- 30-60 minutes

AQUAVEO™

1 Getting Started

Since TUFLOW FV is run through the Generic Model Interface of SMS, we need to load in the TUFLOW FV model definition previously created before we start to create the TUFLOW FV mesh. To do this:

1. Select *File* | *Open* and select the *TUFLOW_FV.2dm* file. This file loads in a model definition.

We will now load in Bathymetry data along with a coverage. To do this:

1. Select *File* | *Open* and select *RiverBend_Bathymetry.tin.* Then, open the *RiverBend_LandUse.map* file.

2. Once the files are read in, select *Display* | *Display Options* and turn in the map option make sure that *Nodes*, *Arcs,* and *Fill* are turned on.

3. In the Scatter option, make sure that *Contours* is turned on and switch to the *Contours* tab. Set the contour method to *Color Fill* and click OK to exit.

4. Click the *Plan view* macro if the objects are not in view.

A scatter set named *RiverBend_Bathymetry* will appear in the Project Explorer along with a new map coverage named *Land_Use.* The *Land_Use* coverage will be read in with its Type set to a 2d Materials TUFLOW Coverage. Figure 1 shows the scatter set and map coverage.

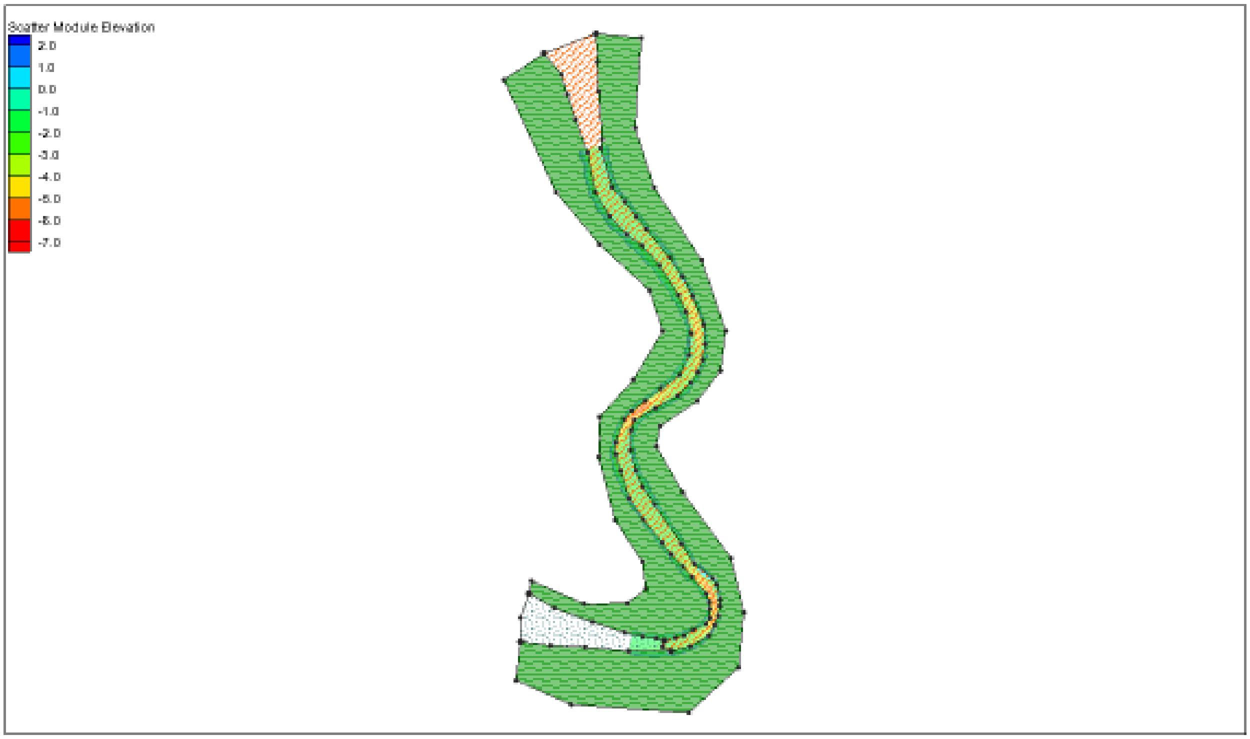


Figure 1

2 Creating the Mesh

Now that we have the required datasets loaded, we can begin creating the model mesh. First, we must create a new coverage in the Map Module, from which we will create our mesh.

1. In the Project Explorer, right-click on the *Map Data* Folder and select *New Coverage.* Set the new coverage's type to *Models* | *Generic 2D Mesh* and set the coverage name to *Mesh_Features.* Click OK when done.

2. We need to define our model extent. Since our model extents will cover our whole bathymetry set, we may use our bathymetry boundaries to define our model extents. To do this, right-click on our *RiverBend_Bathymetry* dataset in the project explorer and select *Convert* | *Scatter Boundary -> Map*

After the conversion, the scatter dataset boundary should be in the Mesh_Features layer (this is easier to see with the scatter set turned off). This is shown in Figure 2.

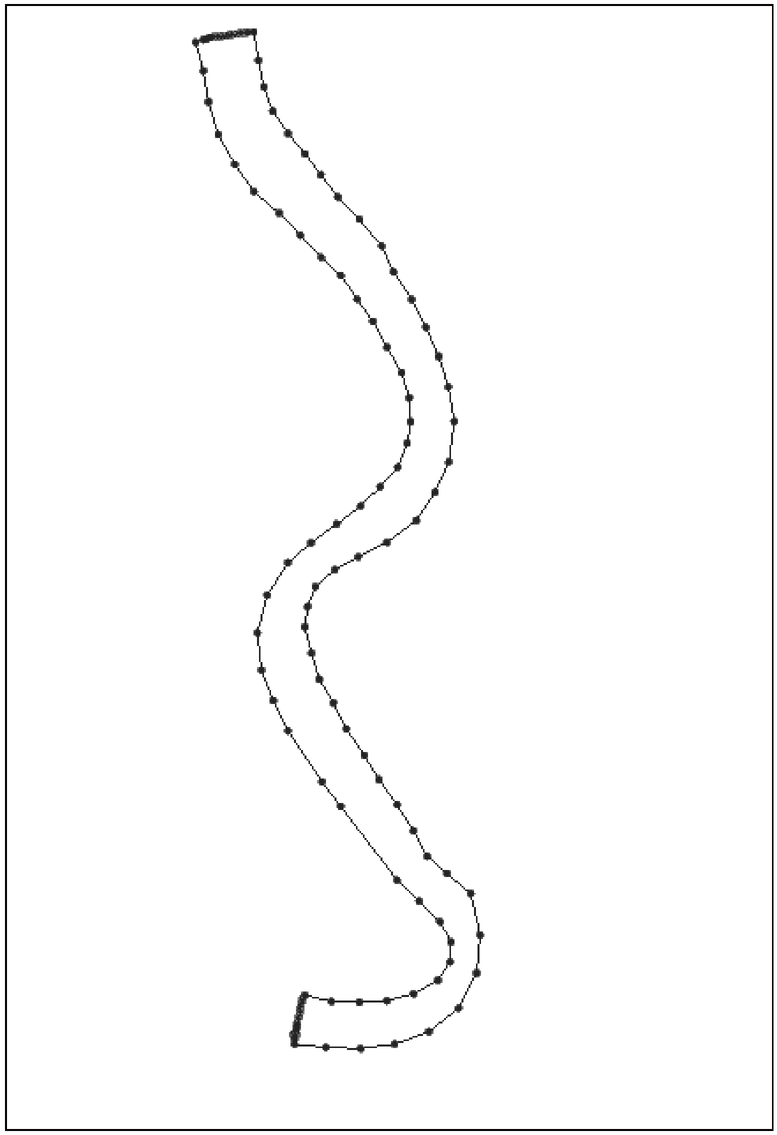


Figure 2

3. Now zoom into the upper boundary of the coverage, and select the two corner vertices using the *Select Feature Vertex*  tool. The two can be selected together by holding down the *Shift* button on your keyboard. Once selected, right-click and select *Convert to Nodes.* Figure 3 shows the nodes that were created.

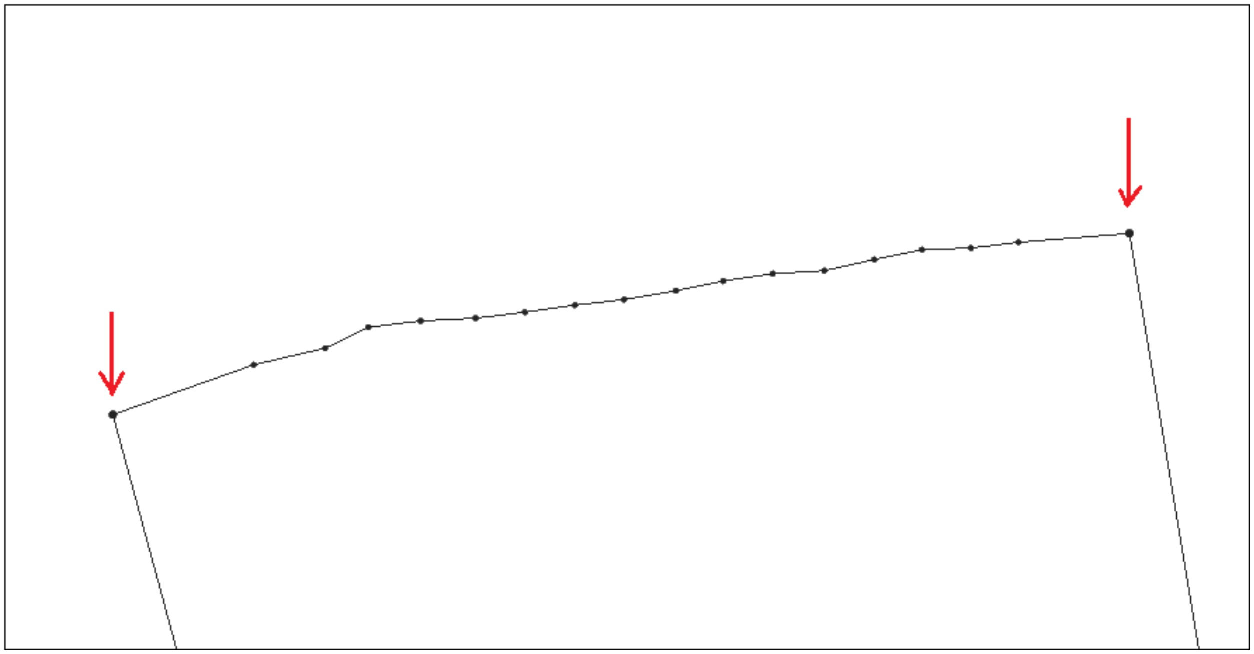


Figure 3

4. Do the same to the lower boundary of the coverage. There is also a node that needs to be converted into a vertex. This is done by using the *Select Feature Point* tool, right-clicking and choosing *Convert to Vertex.* Figure 4 indicates which vertices/nodes need to be converted.

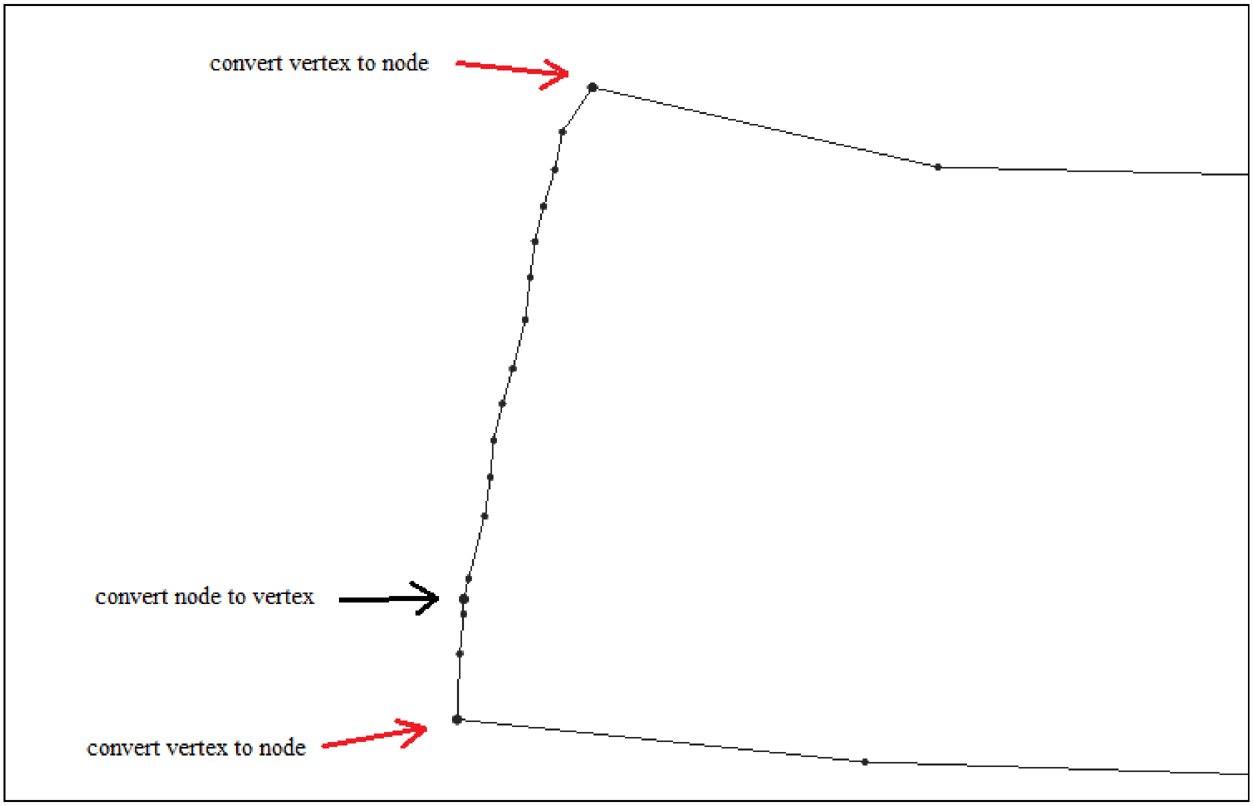


Figure 4

Now we must redistribute the vertices along the arcs to ensure that the mesh can be created. To do this:

1. With the *Select Feature Arc* tool, select the upper arc of the model. Right-click and select *Redistribute Vertices.*

2. Specify the arc redistribution according to *Number of Segments,* and set the Num Seg: as *10.* Leave the bias as 1.0 as shown in Figure 5 and click OK.

3. Repeat the same process for the lower arc in the model

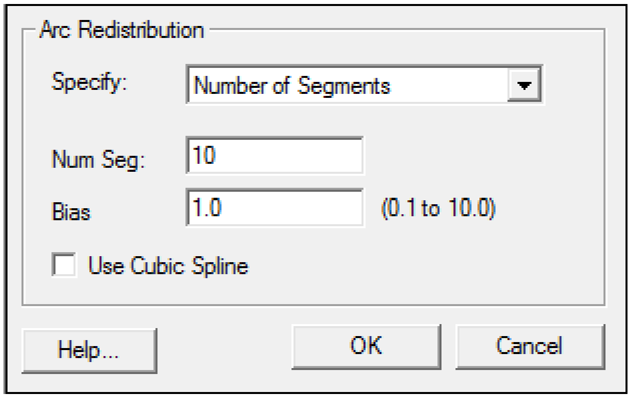


Figure 5

We must redistribute the vertices along the banks of the river as well. To do this:

1. Select both arcs along the banks of the channel with the *Select Feature Arc* tool while holding down *Shift.*

2. Right-click and select *Redistribute Vertices.* For these arcs, we want to specify the arc redistribution as a *Specified Spacing.* Set the spacing as *20.0* meters. Click OK.

In order to build a mesh, polygons must be created from the feature arcs:

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

2. With the *Select Feature Polygon* tool, double click inside the channel. The *2D Mesh Polygon Properties* dialog will appear.

3. Set the Mesh type to *Patch.*

4. Select *Preview Mesh.* You may receive an error about overlapping elements.

In order to avoid this, we should include sections across the channel (perpendicular to flow) at regular spacing along the channel, and particularly around the bends. Create the perpendicular arcs by doing the following:

1. With the *Create Feature Arc* tool, create perpendicular arcs across the channel as shown in Figure 6.

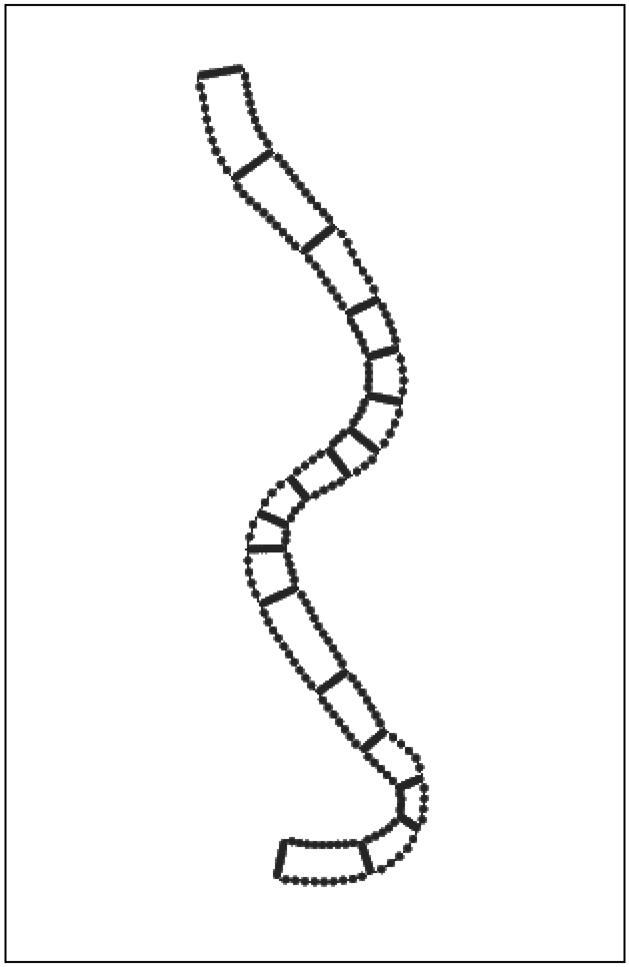


Figure 6

2. Once the section arcs are created, select the *Select Feature Arc* tool and select all of the section arcs while holding down *Shift.* Once selected, right-click and choose *Redistribute Vertices.* Specify the arc redistribution by *Number of Segments* and set the Num Seg: to *10.* Click OK.

With new arcs created, we need to create polygons again.

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

4. With the *Select Feature Polygon* tool, double-click the southern-most polygon. In the *2D Mesh Polygon Properties* dialog select the *Preview Mesh* button.

You may notice that your mesh has both quadrilateral and triangular elements. TUFLOW FV can handle both, but quadrilateral elements are preferred. This occurs because along one bank there are more vertices than the other. To fix this, do the following:

1. In the *2d Mesh Polygon Properties* dialog, select the *Select Feature Arc* tool and select both banks of the channel.

2. Select the *Distribute* option under the Arc Options section and set it to your desired number of vertices. Since both arcs are selected, this will set both arcs to have the same amount of vertices, thus getting rid of any triangular elements. Figure 7 illustrates these steps.

3. Go through all of the polygons and repeat this process to eliminate all triangular elements.

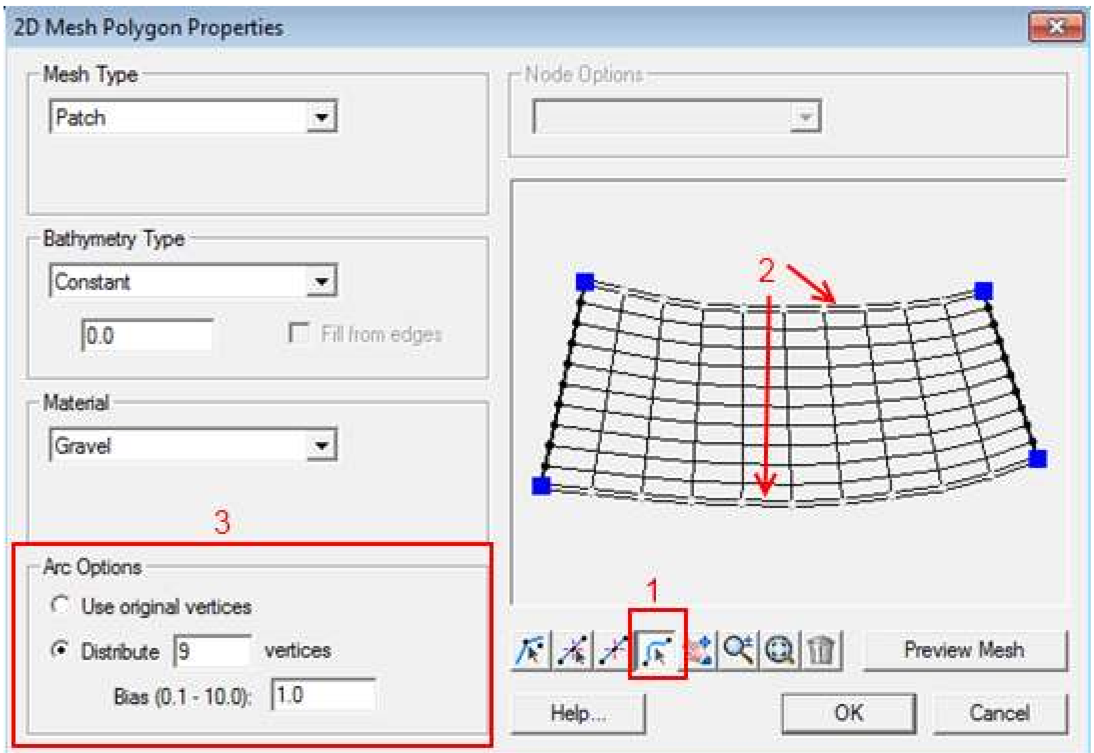


Figure 7

Once you finish going through the model and getting rid of all of the triangular elements, then we must specify an elevation data source for each polygon.

1. Using the *Select Feature Polygon* tool, select all polygons by holding down *Shift.* Once selected, Right-click and select *Attributes.*

2. In the *2D Mesh Multiple Polygon Properties* dialog, Toggle on *Mesh Type* and make sure *Patch* is set for type. Next, toggle on *Bathymetry Type,* and set the type to *Scatter Set.*

3. Click on the *Scatter Options* button and set the extrapolation single value to 2.0 and set *Elevation* as the scatter set to interpolate from. Click OK twice.

3 Setting the Boundary Conditions

This Model will have two boundary conditions: Flow and water level. Assign the boundary conditions by doing the following:

1. Double-click the uppermost arc in the channel using the *Select Feature Arc* tool. In the *Feature Arc Attributes* dialog choose *Boundary Conditions* for the Attribute Type, then select the *Options* button.

2. In the *TUFLOW-FV Nodestring Boundary Conditions* dialog, toggle on *Water Level* and click *Define..*

3. Open the *Tide.csv* file and copy and paste the values into their respective locations in the *XY Series Editor.* All data can be copied at the same time; this does not need to be done one column at a time. Figure 8 shows the *XY Series Editor* dialog. Click OK twice when done.

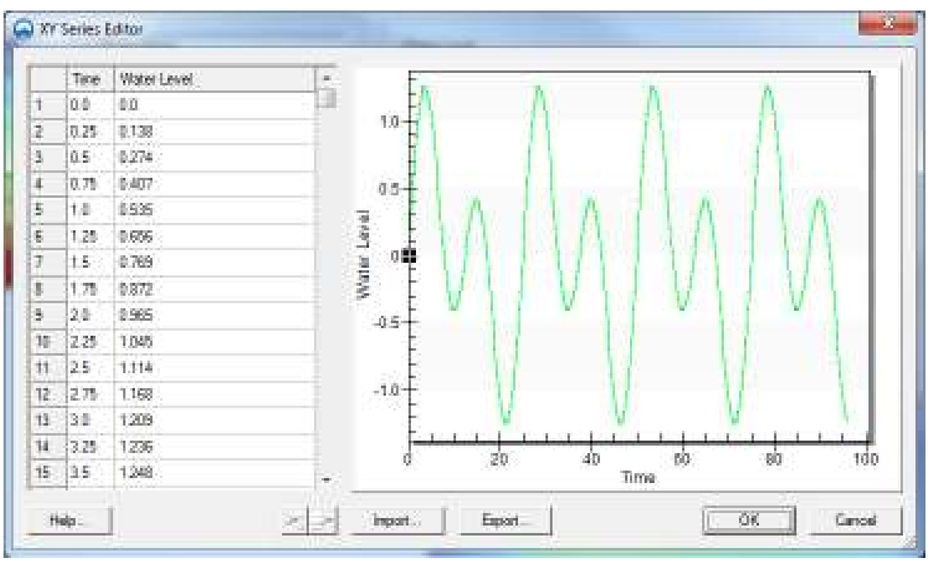


Figure 8

4. Next, select the southernmost arc, double-click and select *Boundary Conditions.* Click on the *Options* button and select *Flow.*

5. Click on *Define*..

6. Open the *Flow.csv* file, then copy and past the values into the *XY Series Editor.* Figure 9 shows the flow data.

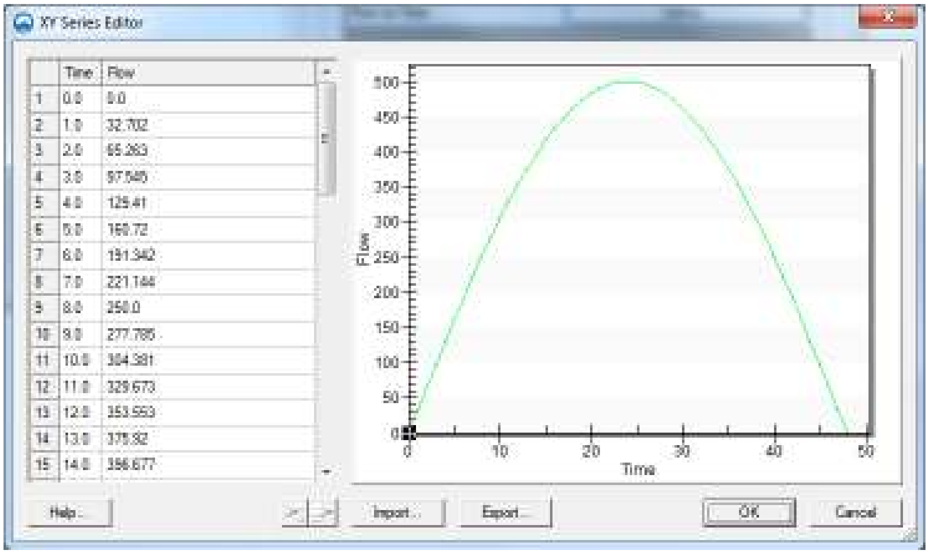


Figure 9

We are now ready to build the mesh from the map data.

1. Select *Feature Objects* | *Map->2D Mesh*

2. In the *2D Mesh Options* dialog, toggle on *Use area coverage.* In the drop box make sure that the *Land Use* Coverage is selected. Click OK to create the mesh.

3. Once the mesh is created, go to *Display* | *Display Options* and with the *2D Mesh* item active, turn on *Elements, Contours* and *Nodestrings.*

4. In the *Contours* Tab, change the contour method to *Color fill,* then click OK.

5. Turn off all of the map data and scatter data in the project explorer, leaving only the mesh data on.

Figure 10 shows what the mesh should look like:

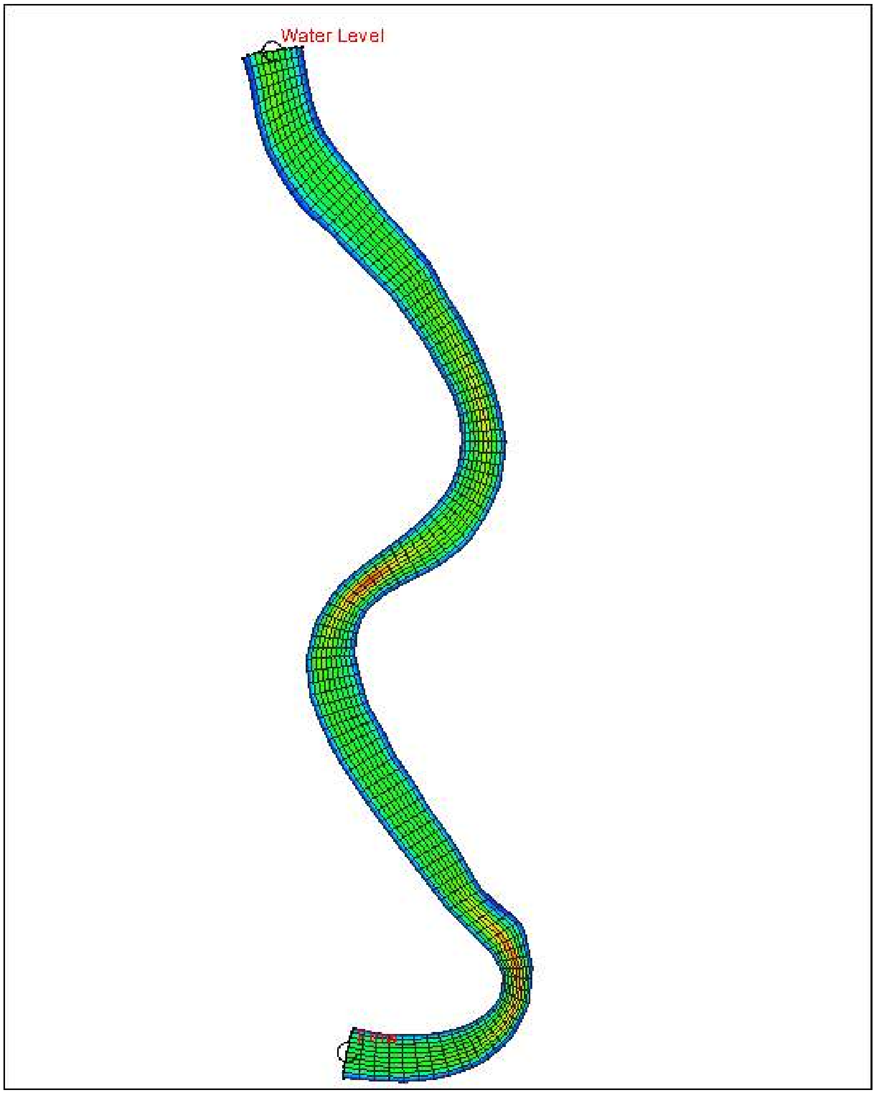


Figure 10

4 Assigning Model Parameters

We will now assign model parameters to our project:

1. Select *TUFLOW FV | Global Parameters..*
2. In the *General* tab, leave the default values.

3. In the *Time* tab, set the Time Format to *Hours,* the Start time to *0,* and the End Time to *48.*

4. In the *Output* tab, make sure the SMS Dat Output is toggled on, and that the Dat Output Types are set to h,v, and that the Dat Output Interval is set to *900.*

5. the *HD Parameters* and the *Advanced Commands* can be left unchanged. Click OK.

5 Setting the Material Properties

Once the global parameters are set, we need to set the Manning's value to be used for the three land types (sand, gravel and vegetated. This section will explain how to do this:

1. Select *TUFLOW-FV | Material Properties..*

2. In the *TUFLOW-FV Material Properties* dialog, set each material to its suggested Manning's value, shown in table 1. Click OK when done.

Land Use	**Suggested Manning's n**
Gravel	0.035
Sand	0.028
Vegetation	0.06

Table 1

6 Saving the Project

Before running the model, we must save our project:

1. Select *File | Save As..* and save the project as *TUFLOW_FV.* Make sure that the file type is .sms and click *Save.*

7 Running the Model

We are now ready to run the model. For this tutorial, we will run the model through a dos prompt with the use of a .bat file. It is important that the TUFLOW FV executable and all of the .dll files associated with it are in the same directory. The .bat file and the mesh_to_FV.exe should also be in the same directory as the TUFLOWFV.exe.

First, copy the .bat and executable files into the same directory that your project is saved in:

1. Select *Edit* | *Preferences* and select the *File Locations* tab.

2. In the Model Executables section, scroll down to the *Generic* model, and click on the directory. Path to where your .bat file is found. Change the *Files of Type* to "All Files (*.*)" and select the *convert_and_run.bat* file, then click *Open.* Click OK to close the *SMS Preferences* dialog.

3. In a file editor such as Notepad ++, open the *convert_and_run.bat* file.

4. Edit lines 7 and 8 so they path to the directory where your executables are found. Figure 11 shows which lines to edit.

```
1  echo off
2  setLocal
3  set input=%1
4  set dir=%cd%
5  set temp=%1:.2dm=.fvc%
6  set output=%input:.2dm=.fvc%
7  set parser="C:\Users\aanderson.AQUAVEO\Documents\TUFLOWFV\mesh_to_FV.exe"
8  set tf_fv=C:\Users\aanderson.AQUAVEO\Documents\TUFLOWFV\TUFLOWFV.exe
9  set fvcpath=%dir%\TUFLOWFV
10
11 echo Aquaveo 1
12 echo Current Directory: %dir%
13 echo Input 2d mesh file: %input%
14 echo Output control file: %output%
15 echo Path to 2dm convertor: %parser%
16 echo Path to TUFLOWFV exe: %tf_fv%
17 echo Press any key to convert to TUFLOW-FV Format
18 echo (where's the any key)
19 pause
20
21 echo Aquaveo 2
22 echo Converting .2dm into .fvc control file:
23 start "convertor" %parser% -b -ow %input%
24 echo Done
25 echo If no errors press any key to start simulation
26 pause
27
28 echo Aquaveo 3
```

Figure 11

5. Once edited, save the .bat file and close it. In SMS, select *TUFLOW-FV | Run TUFLOW-FV*.

A dialog box should come up stating that no model checks have been violated. Click OK

6. A prompt will come up as shown in figure 12 telling you to press any key to continue:

```
c:\users\aanderson.aquaveo\documents\tuflow_fv_run>echo off
Aquaveo 1
Current Directory: c:\users\aanderson.aquaveo\documents\tuflow_fv_run
Input 2d mesh file: TUFLOW_FV.2dm
Output control file: TUFLOW_FV.fvc
Path to 2dm convertor: "C:\Users\aanderson.AQUAVEO\Documents\TUFLOW_FV_Run\mesh_to_FV.exe"
Path to TUFLOWFV exe: C:\Users\aanderson.AQUAVEO\Documents\TUFLOW_FV_Run\TUFLOWFV.exe
Press any key to convert to TUFLOW-FV Format
(where's the any key)
Press any key to continue . . .
```

Figure 12

7. Next, it will again tell you to press any key to continue. This is shown in Figure 13:

```
c:\users\aanderson.aquaveo\documents\tuflow_fv_run>echo off
Aquaveo 1
Current Directory: c:\users\aanderson.aquaveo\documents\tuflow_fv_run
Input 2d mesh file: TUFLOW_FV.2dm
Output control file: TUFLOW_FV.fvc
Path to 2dm convertor: "C:\Users\aanderson.AQUAVEO\Documents\TUFLOW_FV_Run\mesh_to_FV.exe"
Path to TUFLOWFV exe: C:\Users\aanderson.AQUAVEO\Documents\TUFLOW_FV_Run\TUFLOWFV.exe
Press any key to convert to TUFLOW-FV Format
(where's the any key)
Press any key to continue . . .
Aquaveo 2
Converting .2dm into .fvc control file:
Done
If no errors press any key to start simulation
Press any key to continue . . .
```

Figure 13

8. Once you press any key, the model will start running. When the model finishes, it will have written a TUFLOW FV directory to the location where you ran the model.

9. Click on that directory and open the output folder. There will be 2 output files: *TUFLOW_FV_H.dat* and *TUFLOW_FV_V.dat*. Read both be read into SMS.

8 Viewing the Results

Now that the model has run, we can see it once we read in the two output files:

1. Go to *Display | Display Options* and with the *2D Mesh* item make sure *Contours, Elements* and *Vectors.*

2. Switch to the *Contours* tab and make sure the contour method is switched to *Color Fill.*

3. Then switch to the *Vectors* tab and leave everything as the default. Click OK to exit.

4. Click through the new Data sets and scroll through the timesteps.

Figure 14 shows some of the mesh at 1 04:15 with the *TUFLOW_FV_V* Vector data set and the *TUFLOW_FV_H* Scalar data set active.

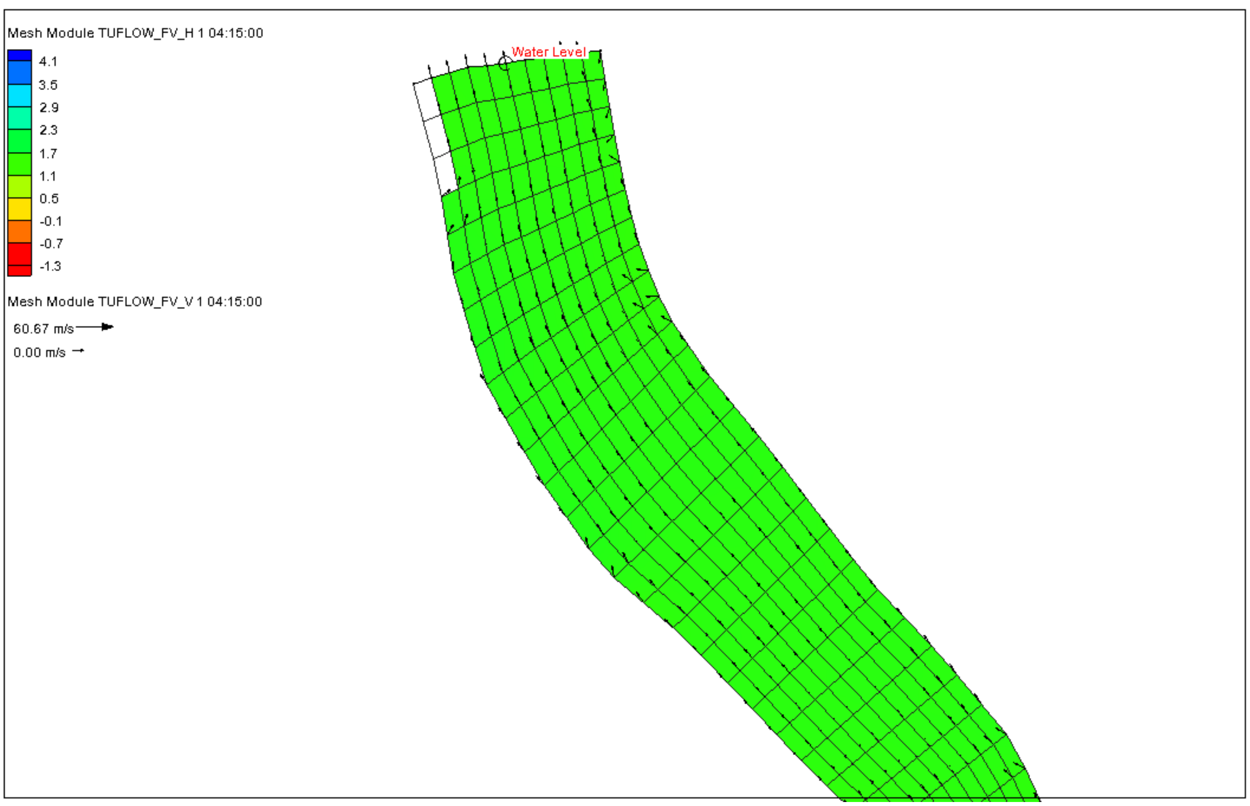


Figure 14

9 Conclusion

This concludes TUFLOW-FV tutorial.