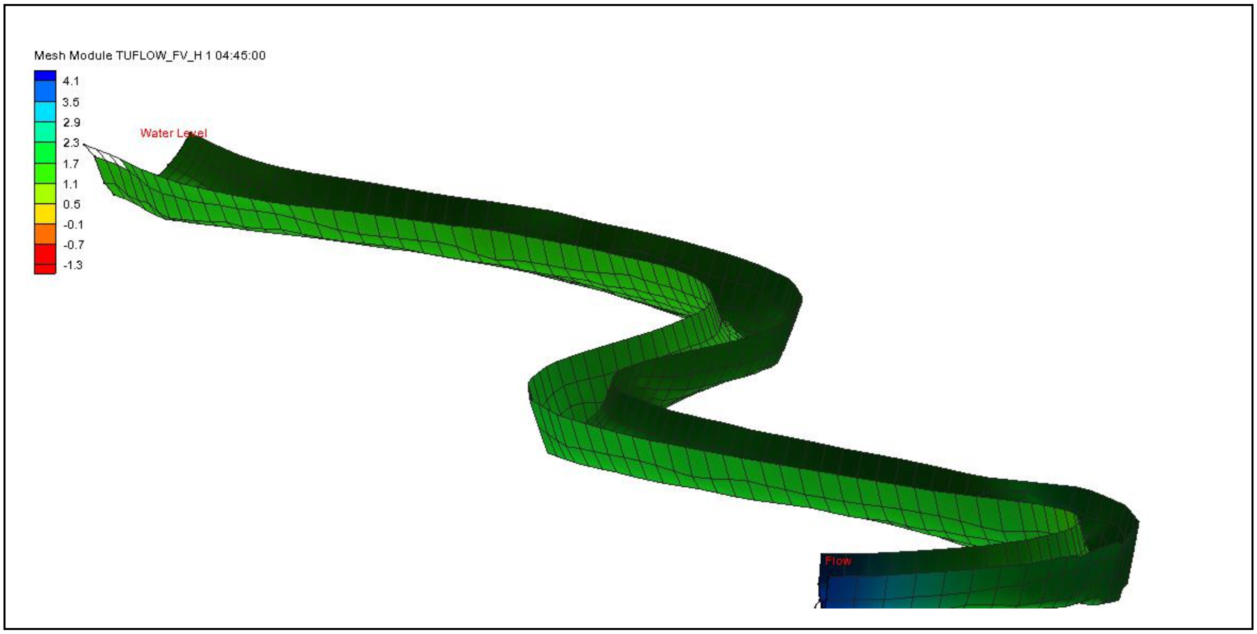
v. 12.1

SMS 12.1 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. A mesh for an inbank area of a river will be built, and an upstream inflow boundary and a downstream tidal boundary will be applied.

Prerequisites

- None

Requirements

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

Time

- 30-60 minutes

1 Getting Started

Since TUFLOW FV is run through the Generic Model Interface of SMS, the TUFLOW FV model definition previously created must be imported before starting to create the TUFLOW FV mesh.

Import the definition and the bathymetry data with a coverage by doing the following:

1. Select *File* | **Open** and browse to the *data files* directory.
2. Select “TUFLOW_FV.2dm” and click the **Open** button. This file imports a model definition, though nothing visibly happens in SMS.
3. Select *File* | **Open**, select “RiverBend_Bathymetry.tin”, and click the **Open** button.
4. Select *File* | **Open**, select “RiverBend_LandUse.map”, and click the **Open** button.
5. Select *Display* | **Display Options** to bring up the *Display Options* dialog and highlight *Map* from the list on the left.
6. Toggle on *Nodes*, *Arcs*, and *Fill*.
7. Select *Scatter* from the list on the left and toggle on *Contours*.
8. On the *Contours* tab, set the *Contour method* to “Color Fill”.
9. Click **OK** to exit the *Display Options* dialog.
10. Click the **Frame** 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 was imported with its *Type* set to a “2D Materials TUFLOW Coverage”. Figure 1 shows the scatter set and map coverage.

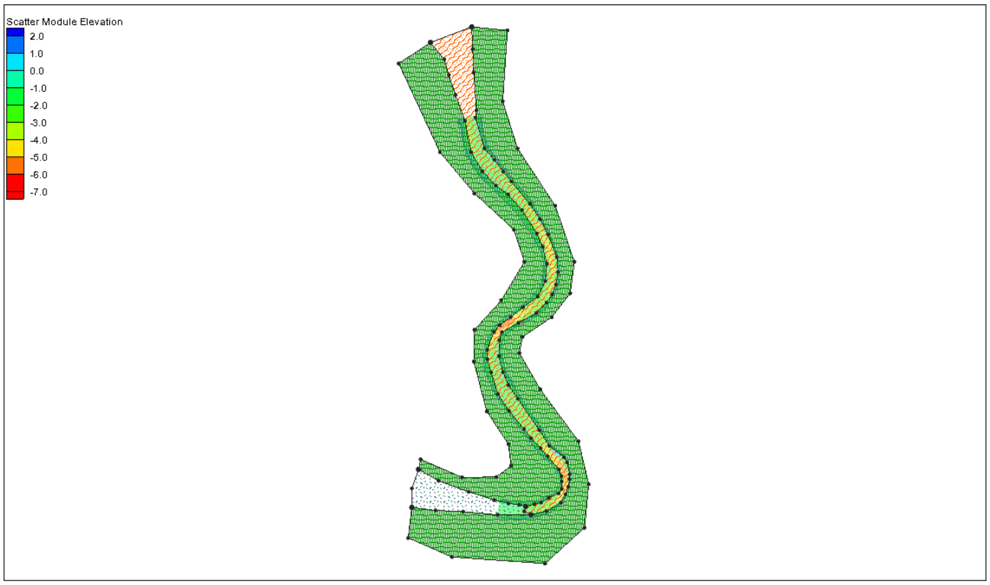


Figure 1 Scatter set and map coverage

2 Creating the Mesh

Now that the required datasets are loaded, the model mesh can be created. This is done by first creating a new coverage in the Map Module.

To do this:

1. In the Project Explorer, right-click on the *Map Data* folder and select **New Coverage**.
2. In the *New Coverage* dialog, set *Coverage Type* to *Model* | **Generic Model.**
3. Set *Coverage Name* to "Mesh_Features".
4. Click **OK** to close the *New Coverage* dialog.

Since the model extents will cover the whole bathymetry set, the bathymetry boundaries may be used to define the model extents by doing the following:

5. Right-click on the "RiverBend_Bathymetry" dataset in the Project Explorer and select *Convert* | ***Scatter Boundary*** → ***Map*** to bring up the *Select Coverage* dialog.
6. Select the *Use existing coverage* radio button.
7. Click on the **Select…** button to bring up the *Select Tree Item* dialog.
8. Select "Mesh_Features" from the tree.
9. Click **OK** to close the *Select Tree Item* dialog and **OK** again to close the *Select Coverage* dialog.

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 (Figure 2).

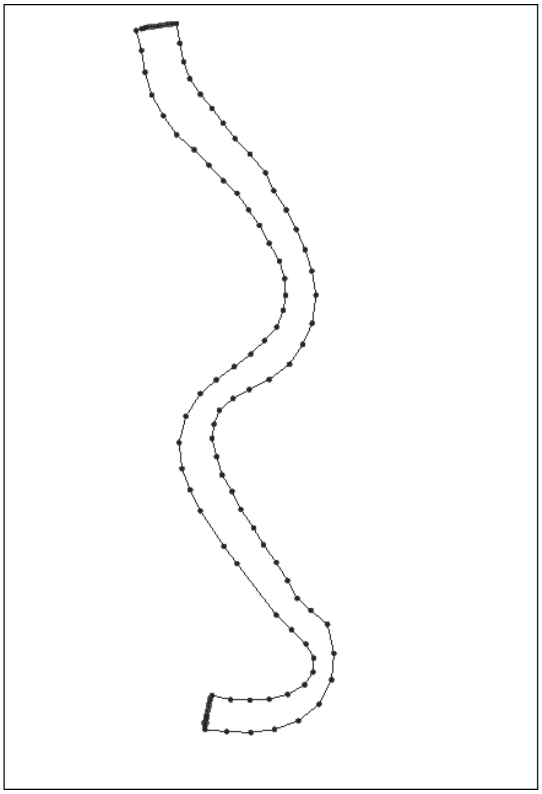


Figure 2 Scatter dataset boundary with the scatter set turned off

10. **Zoom** into the upper boundary of the coverage as shown in Figure 3.
11. Select the two corner vertices using the **Select Feature Vertex** tool. The two can be selected together by holding down the *Shift* key.
12. Once selected, right-click and select **Convert to Nodes**. Figure 3 shows the nodes that were created.

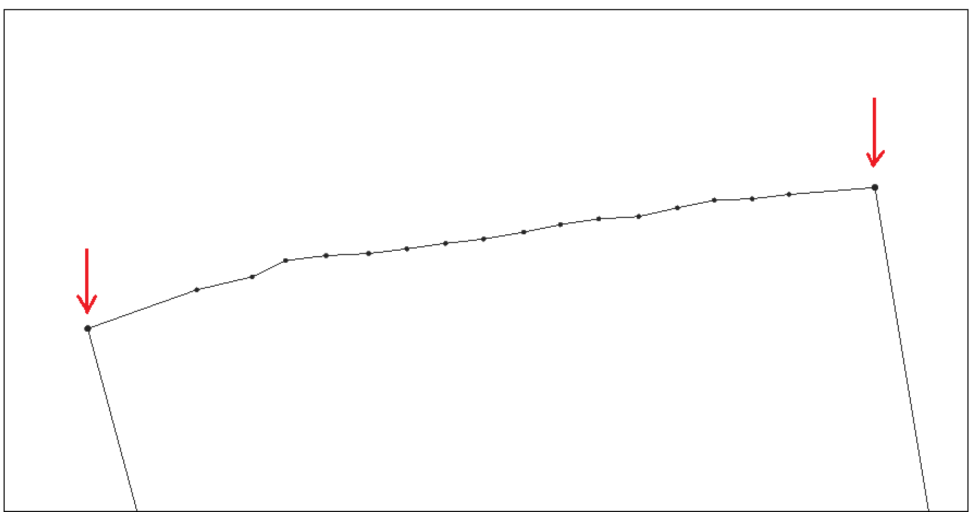


Figure 3 Vertices converted to nodes

13. Do the same to the lower boundary of the coverage (see red arrows in Figure 4).

There is also a node that needs to be converted into a vertex.

14. Using the **Select Feature Point** tool, select the node on the lower boundary arc.
15. Right-click and choose **Convert to Vertex** (see black arrow in Figure 4).

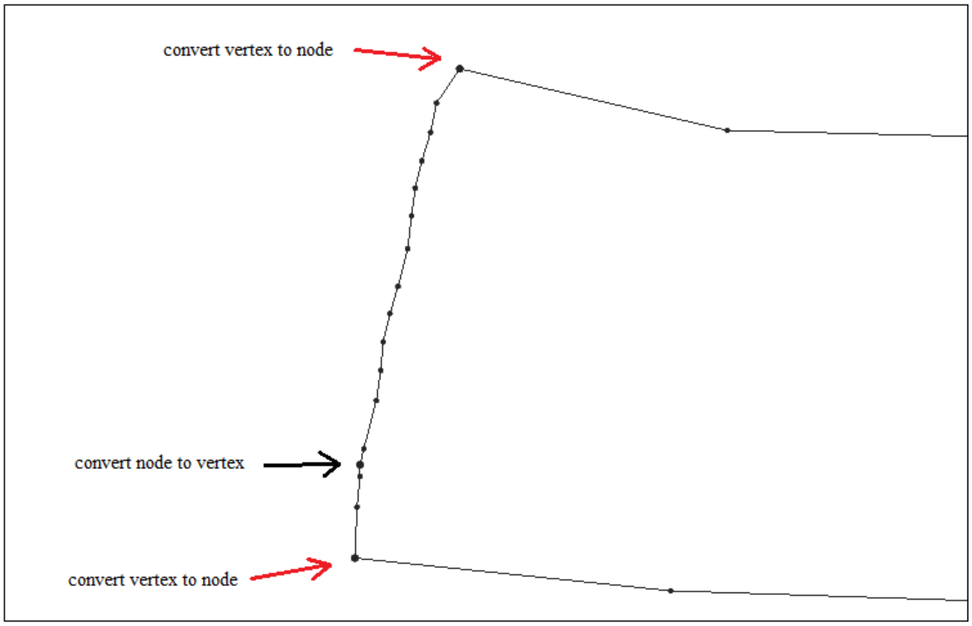


Figure 4 Nodes to be converted to vertices

Now the vertices must be redistributed along the arcs to ensure that the mesh can be created.

To do this:

1. With the **Select Feature Arc** tool, select the upper inflow arc of the model.
2. Right-click and select **Redistribute Vertices** to bring up the *Redistribute Vertices* dialog.
3. In the *Arc Redistribution* section, set *Specify* to “Number of Segments”.
4. Set *Number of* to “10”. Leave everything else at the default settings (Figure 5).
5. Click **OK** to close the *Redistribute Vertices* dialog.
6. Repeat the same process for the lower outflow arc in the model.

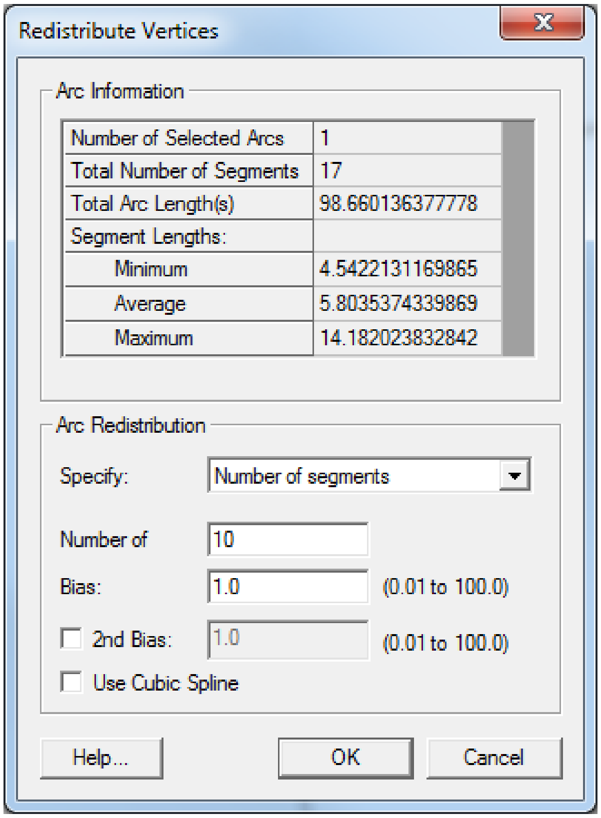


Figure 5 Arc Redistribution dialog

The vertices along the banks of the river must also be redistributed. To do this:

1. Select both arcs along the banks of the channel with the **Select Feature Arc** tool while holding down the *Shift* key.
2. Right-click and select **Redistribute Vertices** to bring up the *Redistribute Vertices* dialog again.
3. Set *Specify* to “Specified Spacing”.
4. Set *Average* to “20.0”.
5. Click **OK** to close the *Redistribute Vertices* dialog.

Polygons must be created from the feature arcs in order to build a mesh. To do this:

1. Select *Feature Objects* | **Build Polygons.**
2. With the **Select Feature Polygon** tool, double-click inside the channel to bring up the *2D Mesh Polygon Properties* dialog.
3. Set *Mesh type* to “Patch”.
4. Click the **Preview Mesh** button. An error about overlapping elements may appear.
5. Click **OK** to close the error message.
6. Click **Cancel** to close the *2D Mesh Polygon Properties* dialog.

In order to avoid overlapping elements, perpendicular arcs should be created across the channel at regular spacing along the channel, and particularly around the bends by doing the following:

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

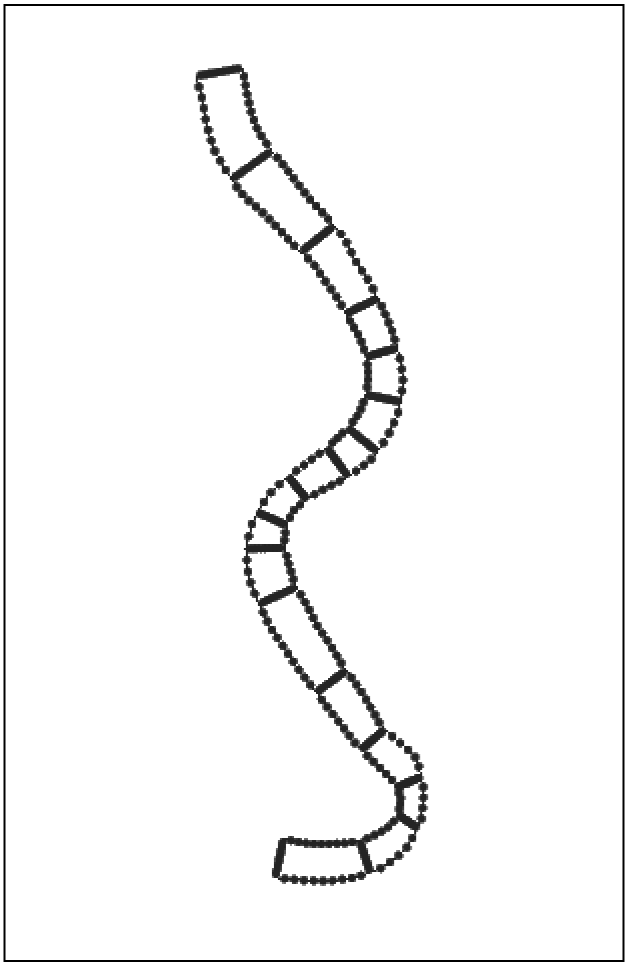


Figure 6 Arcs created across the channel

2. Select the **Select Feature Arc** tool and select all of the section arcs while holding down the *Shift* key.
3. Once selected, right-click and choose **Redistribute Vertices** to bring up the *Redistribute Vertices* dialog.
4. Set *Specify* to “Number of Segments”.
5. Set the *Number of* to “10”.
6. Click **OK** to close up the *Redistribute Vertices* dialog.

With new arcs created, polygons must again be created:

1. Select *Feature Objects* | **Build Polygons**.
2. With the **Select Feature Polygon** tool, double-click the southern-most polygon to bring up the *2D Mesh Polygon Properties* dialog.

3. Change the *Mesh Type* to “Patch”.
4. Select the **Preview Mesh** button.

At this point, the mesh may have both quadrilateral and triangular elements. TUFLOW FV can handle both, but quadrilateral elements are preferred. This occurs because the banks do not have an equal number of vertices.

To fix this, do the following:

5. Click on the **Select Feature Arc** tool in the dialog (labelled “1” in Figure 7).
6. Select both banks of the channel using the *Shift* key (labelled “2” in Figure 7).
7. Select the *Distribute* option under the *Arc Options* section and set it to the default number of suggested vertices (or to any other reasonable number).
8. Click the **Preview Mesh** button. Both arcs will now have the same amount of vertices, removing any triangular elements (labelled “3” in Figure 7).
9. Click **OK** to close the *2D Mesh Polygon Properties* dialog.
10. Repeat steps 2-9 for all of the polygons to eliminate all triangular elements.

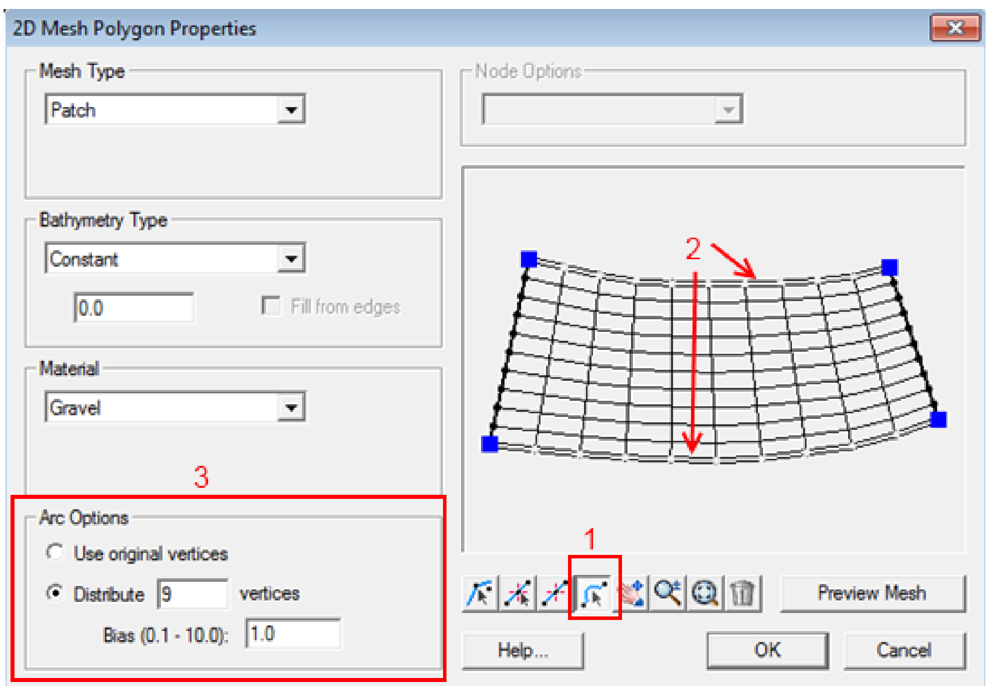


Figure 7 Setting the arc options in the 2D Mesh Polygon Properties dialog

An elevation data source must now be specified for each polygon, as follows:

1. Using the **Select Feature Polygon** tool, select all polygons by holding down *Shift*.
2. Right-click and select **Attributes** to bring up the *2D Mesh Multiple Polygon Properties* dialog.
3. Turn on *Mesh Type* and set it to “Patch”.
4. Turn on *Bathymetry Type*, and set it to “Scatter Set”.

5. Click on the **Scatter Options** button to bring up the *Interpolation* dialog.
6. Set Extrapolation to "Single Value".
7. Set *Single Value* to "2.0".
8. Select "Elevation" in the *Scatter Set To Interpolate From* section.
9. Click **OK** to close the *Interpolation* dialog and click **OK** again to close the *2D Mesh Multiple Polygon Properties* dialog.

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 inflow (top) arc in the channel using the **Select Feature Arc** tool to bring up the *Feature Arc Attributes* dialog.
2. Select *Boundary Conditions* in the *Attribute Type* section.
3. Click on the **Options** button to bring up the *TUFLOW-FV Nodestring Boundary Conditions* dialog.
4. Toggle on *Water Level* in the tree on the left.
5. Click the **Define** button in the *Value* column next to *water level vs time* in the *Water Level* section on the right. This brings up the *XY Series Editor* dialog.
6. Open the file "Tide.csv" in a spreadsheet program.
7. Copy the time values from the *x* column in the "Tide.csv" file to the *Time* column in the *XY Series Editor* dialog.
8. Copy the flow values from the *y* column in the "Tide.csv" file to the *Water Level* column in the *XY Series Editor* dialog. See Figure 8.
9. Click **OK** to close the XY Series Editor.
10. Click **OK** to close the *TUFLOW-FV Nodestring Boundary Conditions* dialog.
11. Click **OK** to close the *Feature Arc Attributes* dialog.

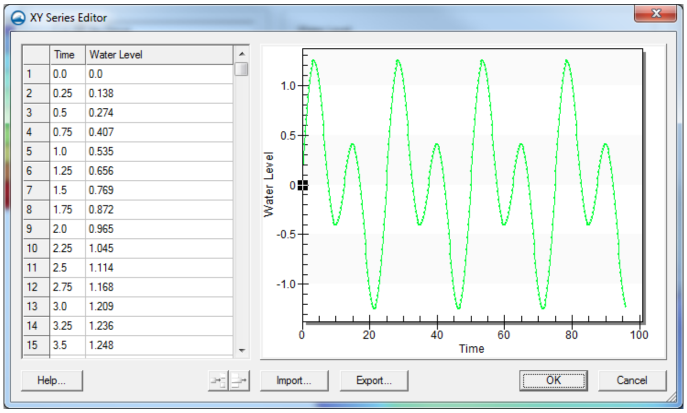


Figure 8 The dialog as it should appear once the tide values are entered

12. Double-click the outflow (bottom) arc in the channel using the **Select Feature Arc** tool to bring up the *Feature Arc Attributes* dialog.
13. Select *Boundary Conditions* in the *Attribute Type* section.
14. Click on the **Options** button to bring up the *TUFLOW-FV Nodestring Boundary Conditions* dialog.
15. Toggle on *Flow* in the tree on the left.
16. Click the **Define** button in the *Value* column next to *Flow vs time* in the *Flow* section on the right. This brings up the *XY Series Editor* dialog.
17. Open the file "flow.csv" in a spreadsheet program.
18. Copy the time values from the *x* column in the "flow.csv" file to the *Time* column in the *XY Series Editor* dialog.
19. Copy the flow values from the *y* column in the "flow.csv" file to the *Flow* column in the *XY Series Editor* dialog. See Figure 9.
20. Click **OK** to close the XY Series Editor.
21. Click **OK** to close the TUFLOW-FV Nodestring Boundary Conditions dialog.
22. Click **OK** to close the *Feature Arc Attributes* dialog.

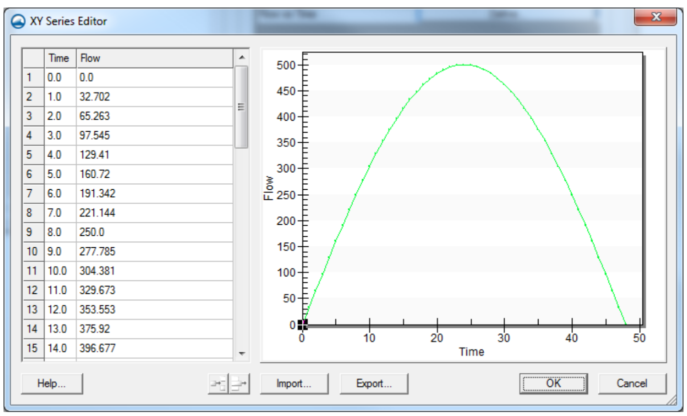


Figure 9 The dialog as it should appear once the flow values are entered

To build the mesh from the map data, take the following steps

1. Select *Feature Objects* | **Map→2D Mesh** to bring up the *2D Mesh Options* dialog.
2. Turn on *Use area coverage*.
3. Select "Land Use" in the drop-box under *Use area coverage*.
4. Click **OK** to create the mesh.
5. Click **OK** when a message states how many elevations were extrapolated.
6. Accept the default *Mesh name* in the *Mesh Name* dialog and click **OK**.
7. Select *Display* | **Display Options** to bring up the *Display Options* dialog and select "2D Mesh" from the list on the left.
8. Toggle on *Elements*, *Contours*, and *Nodestrings*.

9. On the *Contours* tab, change the *Contour method* to “Color Fill”.
10. Click **OK** to close the *Display Options* dialog.
11. Turn off Map Data and Scatter Data in the Project Explorer, leaving only Mesh Data visible. Figure 10 shows what the mesh should look like:

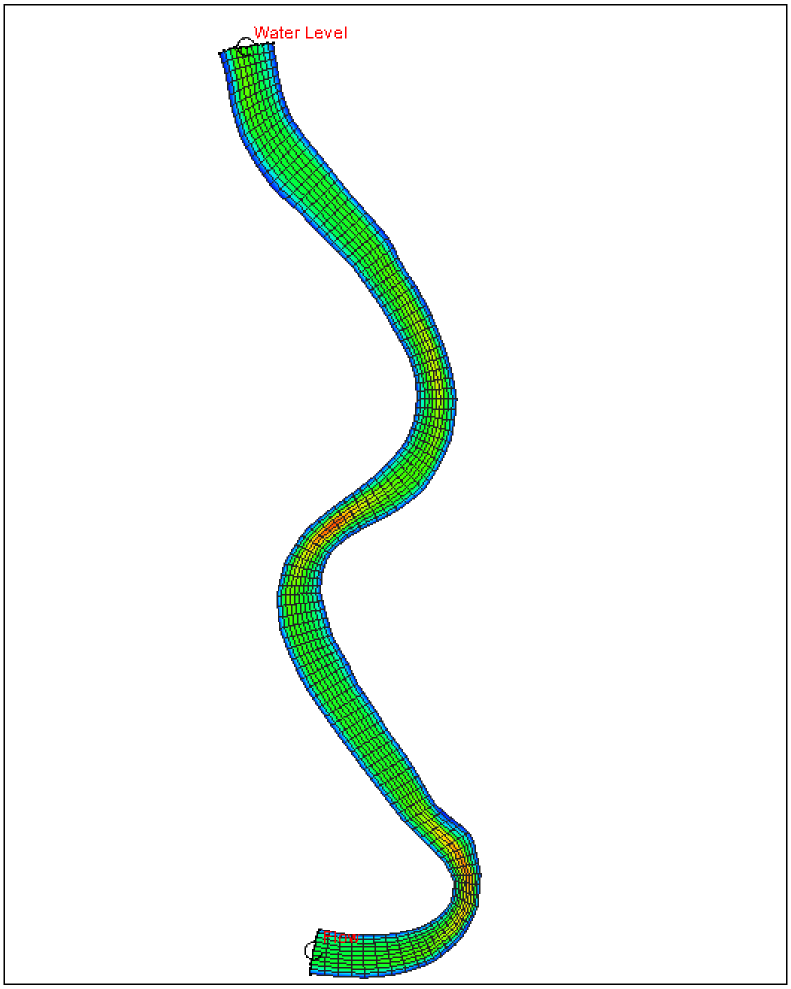


Figure 10 The channel with only mesh data visible

4 Assigning Model Parameters

Assign model parameters to the project using the following steps:

1. Select “Mesh_Features Mesh” in the Project Explorer to make it active.
2. Select *TUFLOW FV* | **Global Parameters** to bring up the *TUFLOW-FV Global Parameters* dialog.
3. Switch to the *Time* tab and set:
   - *Time Format* to “Hours”
   - *Start time* to “0”.
   - *End Time* to “48”.

4. Switch to the *Output* tab and do the following:
   - Turn on *SMS Dat Output*.
   - Set *Dat Output Types* to “h,v”.
   - Set *Dat Output Interval* to “900”.
5. The settings on the *General*, *HD Parameters*, and *Advanced Commands* tabs can be left unchanged.
6. Click **OK** to close the *TUFLOW-FV Global Parameters* dialog.

5 Setting the Material Properties

Once the global parameters are set, set the Manning's *n* value to be used for each of the three land types (sand, gravel and vegetated) by doing the following:

1. Select *TUFLOW-FV* | **Material Properties...** to bring up the *TUFLOW-FV Material Properties* dialog.
2. Set each material to the Manning's *n* value shown in the table below.
3. Click **OK** when done to close the *TUFLOW-FV Material Properties* dialog.

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

6 Saving the Project

Before running the model, save the project:

1. Select *File* | **Save As...** and set the *File name* as “TUFLOW_FV.sms”.
2. Click **Save**.

7 Running the Model

For this tutorial, 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 “mesh_to_FV.EXE” should also be in the same directory as “TUFLOWFV.EXE”.

Copy all of the BAT, DLL, and EXE files into the project directory:

1. Browse to the *models\TUFLOWFV\x64* (or *models\TUFLOWFV\win32* if the 32-bit version is installed) directory within the directory where SMS is installed.
2. Select all the files (*Ctrl-A*) and copy the files (*Ctrl-C*).
3. Browse to the directory where the project is saved and paste the files (*Ctrl-V*).

4. Select *Edit* | **Preferences** to bring up the *SMS Preferences* dialog.
5. Select the *File Locations* tab.
6. In the *Model Executables* section, scroll down to the “Generic” model, and click on the directory path (it may be titled “BROWSE”) to where the BAT file is found.
7. Change the *Files of type* to "All Files (*.*)" and browse to the project directory.
8. Select “convert_and_run.bat” and click **Open**.
9. Click **OK** to close the *SMS Preferences* dialog.

Next, edit the BAT file to point to the correct directory:

1. In a file editor such as *Notepad ++*, open the “convert_and_run.bat” file.
2. Edit the line starting with “set parser=” so it points to the project directory (*data files*) where “mesh_to_FV.exe” is found.
3. Edit the line starting with “set tf_fv=” so it points to the project directory where “TUFLOWFV.exe” is found (the same directory as in step 8, above). See Figure 11 for what the lines should look like.

```
echo off
setLocal
set input=%1
set dir=%cd%
set temp=%1:.2dm=.fvc%
set output=%input:.2dm=.fvc%
set parser="C:\Users\aanderson.AQUAVEO\Documents\TUFLOWFV\mesh_to_FV.exe"
set tf_fv=C:\Users\aanderson.AQUAVEO\Documents\TUFLOWFV\TUFLOWFV.exe
set fvcpath=%dir%\TUFLOWFV

echo Aquaveo 1
echo Current Directory: %dir%
echo Input 2d mesh file: %input%
echo Output control file: %output%
echo Path to 2dm convertor: %parser%
echo Path to TUFLOWFV exe: %tf_fv%
echo Press any key to convert to TUFLOW-FV Format
echo (where's the any key)
pause

echo Aquaveo 2
echo Converting .2dm into .fvc control file:
start "convertor" %parser% -b -ow %input%
echo Done
echo If no errors press any key to start simulation
pause

echo Aquaveo 3
```

Figure 11 The "set parser" and "set tf_fv" lines need to be edited

4. Once edited, save the BAT file and close it.
5. Insert the TUFLOW-FV dongle into a free USB port on the computer.
6. Go back to SMS and select *TUFLOW-FV* | **Run TUFLOW-FV**.
7. A dialog box will advise that no model checks have been violated. Click **OK**.
8. A prompt will ask for any key to be pressed to continue. Press any key on the keyboard to continue. Repeat this step when the prompt appears a second time.

The model may take several minutes to finish running. When the model finishes, it will have written a *TUFLOWFV* directory to the location where the model was run.

9. Use the *File* | **Open** command and browse to the *output* folder within the *TUFLOWFV* directory.
10. Open the two files contained within that directory: *"*TUFLOW_FV_H.dat" and "TUFLOW_FV_V.dat". This will add additional datasets in the Mesh Data folder in the Project Explorer.

8 Viewing the Results

Now that the model has run, the results can be viewed in SMS:

1. Select *Display* | **Display Options** to bring up the *Display Options* dialog and select "2D Mesh" from the list on the left.
2. Toggle on *Contours*, *Elements* and *Vectors*.
3. On the *Contours* tab, set *Contour method* to "Color Fill".
4. Click **OK** to exit the *Display Options* dialog.
5. Click through the new datasets and scroll through the time steps.

Figure 12 shows some of the mesh at 1 04:15 with the *TUFLOW_FV_V* Vector dataset and the *TUFLOW_FV_H* Scalar dataset active.

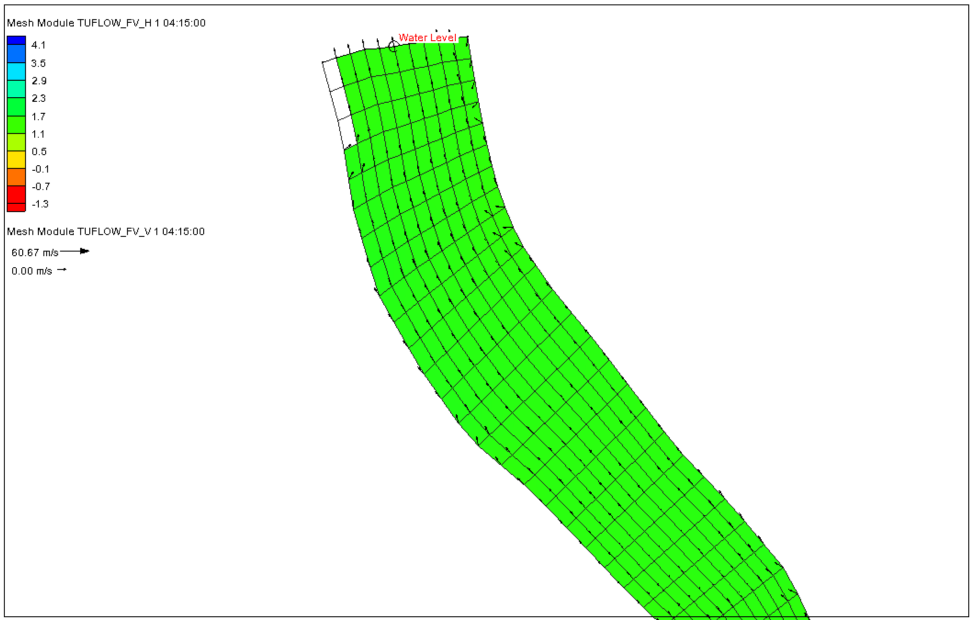


Figure 12   A portion of the mesh with TUFLOW_FV_V and TUFLOW_FV_H active

9 Conclusion

This concludes the TUFLOW-FV tutorial. End the program or continue experimenting.