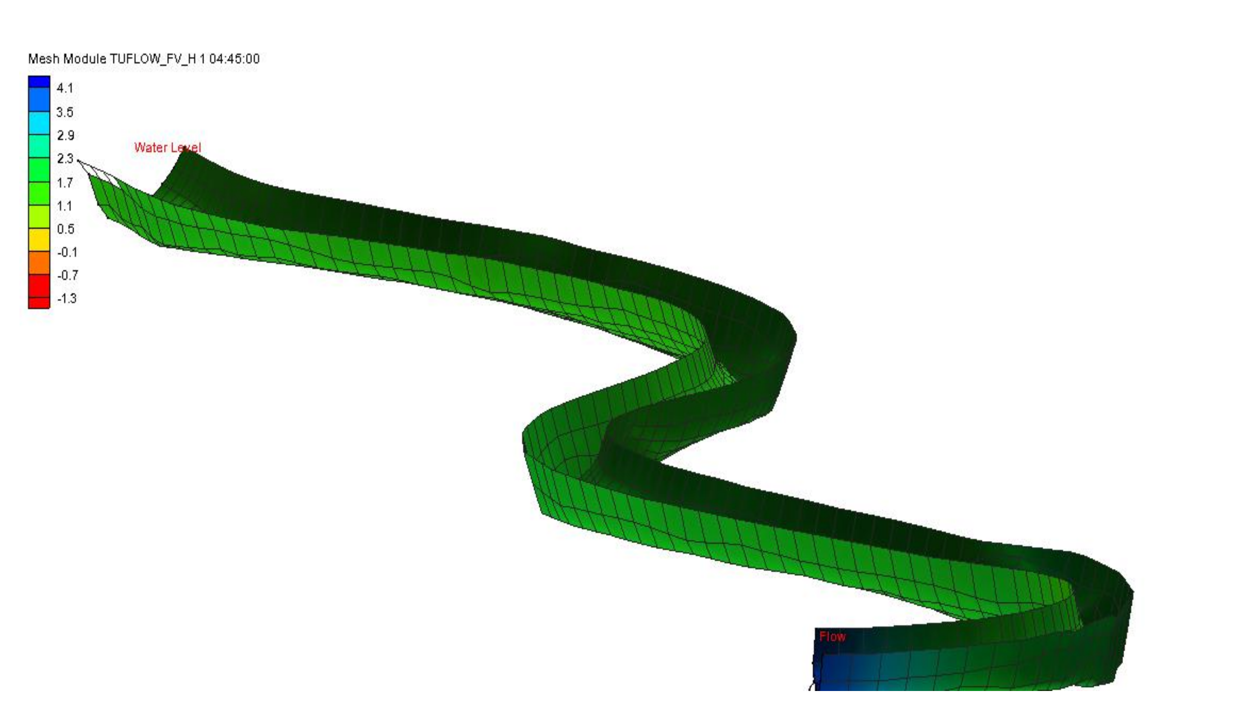
SMS 12.3 Tutorial

TUFLOW FV



Objectives

This tutorial demonstrates creating a simple model of a short section of river 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

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.

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…** to bring up the *Open* dialog.
2. Browse to the *Data Files* directory for this tutorial and select "TUFLOW_FV.2dm".
3. Click **Open** to import the 2DM file and exit the *Open* dialog.

This file imports a only model definition, so nothing visibly happens in SMS.

4. Select *File* | **Open…** to bring up the *Open* dialog.
5. Select "RiverBend_Bathymetry.tin" and click **Open** to import the TIN and exit the *Open* dialog.
6. Click **Open** to bring up the *Open* dialog.
7. Select "RiverBend_LandUse.map" and click **Open** to import the map file and exit the *Open* dialog.
8. Select *Display* | **Display Options…** to bring up the *Display Options* dialog.
9. Select "Map" from the list on the left.
10. Turn on *Node*, *Arc*, *Fill*, and *Legend*.
11. Select "Scatter" from the list on the left.
12. On the *Scatter* tab, turn on *Contours*.

13. On the *Contours* tab, in the *Contour method* section, select "Color Fill" from the first drop-down.
14. Click **OK** to exit the *Display Options* dialog.
15. **Frame** the project.

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 "2D Materials TUFLOW Coverage". The project should appear similar to Figure 1.

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.

2.1 Creating a New Coverage

Create the new coverage by doing the following:

1. Right-click on " Map Data" in the Project Explorer and select **New Coverage** to bring up the *New Coverage* dialog.
2. In the *Coverage Type* section, select *Model* | **Generic Model**.
3. Enter "Mesh Features" as the *Coverage Name*.
4. Click **OK** to close the *New Coverage* dialog.

2.2 Setting Up the Boundary Arcs

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:

1. Right-click on " RiverBend_Bathymetry" and select *Convert* | **Scatter Boundary → Map** to bring up the *Select Coverage* dialog.
2. Select *Use existing coverage* and click **Select…** to bring up the *Select Tree Item* dialog.
3. Select "Mesh_Features" from the tree list and click **OK** to close the *Select Tree Item* dialog.
4. Click **OK** to close the *Select Coverage* dialog.
5. Turn off " RiverBend_Bathymetry".

The project should appear similar to Figure 2.

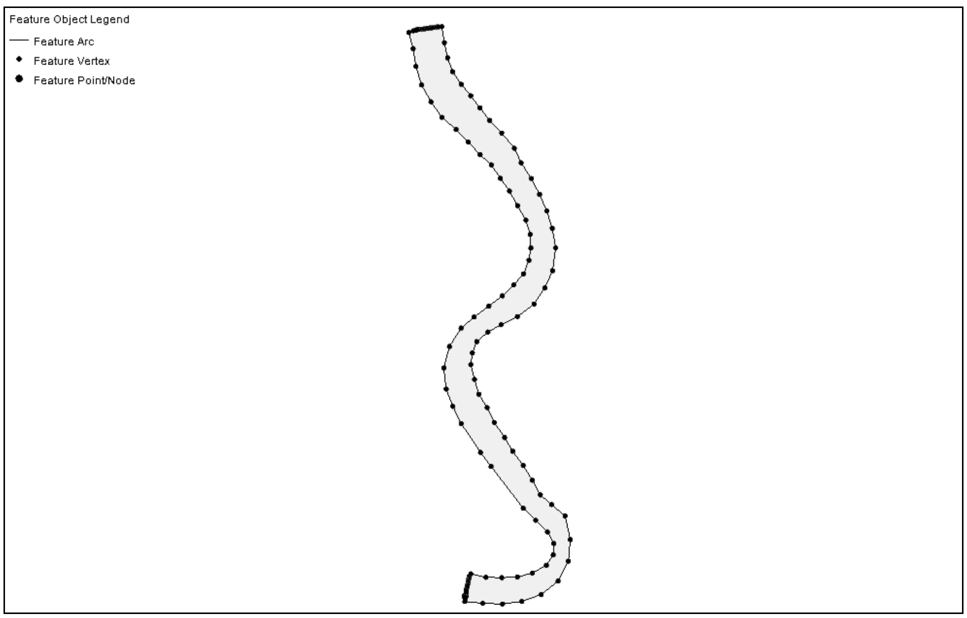


Figure 2 Scatter dataset boundary with the scatter set turned off

6. **Zoom** into the upper boundary of the coverage as shown in Figure 3.
7. Using the **Select Feature Vertex** tool while holding down the *Shift* key, select the two corner vertices indicated in Figure 3.
8. Right-click and select **Convert to Nodes**.

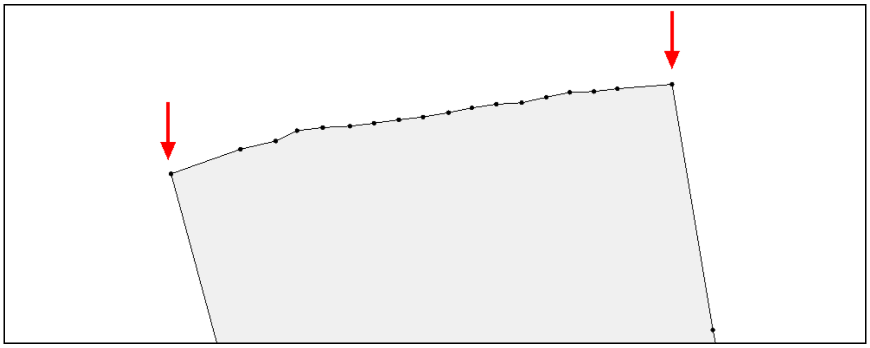


Figure 3 Vertices converted to nodes

9. **Zoom** in to the lower boundary of the coverage as shown in Figure 4.
10. Repeat steps 7–8 for the two corner vertices indicated in Figure 4.

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

11. Using the **Select Feature Point** tool, select the node three up from the newly-created bottom corner node as indicated in Figure 4.
12. Right-click and select **Convert to Vertex**.

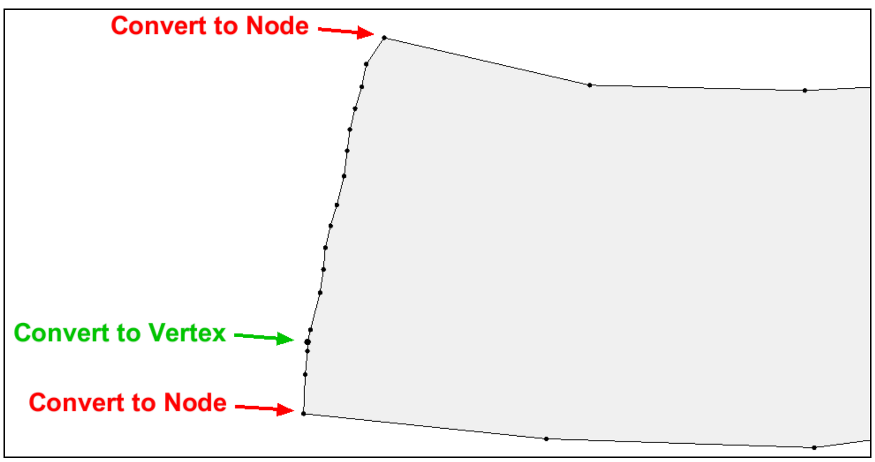


Figure 4 Nodes to be converted to vertices

Now the vertices must be redistributed along the inflow, outflow, and bank boundary arcs to ensure that the mesh can be created.

To redistribute the vertices along the inflow and outflow arcs, do the following:

1. **Frame** the project, then **Zoom** in to the upper inflow arc.
2. Using the **Select Feature Arc** tool, select the upper inflow arc of the model.
3. Right-click and select **Redistribute Vertices…** to bring up the *Redistribute Vertices* dialog.
4. In the *Arc Redistribution* section, select “Number of segments” from the *Specify* drop-down.
5. Enter “10” as the *Number of segments*.
6. Click **OK** to close the *Redistribute Vertices* dialog.

7. Repeat steps 1–6 for the lower outflow arc in the model.

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

8. Deselect the lower outflow boundary arc by clicking anywhere other than an arc.
9. Using the **Select Feature Arc** tool while holding down the *Shift* key, select both channel bank arcs.
10. Right-click and select **Redistribute Vertices…** to bring up the *Redistribute Vertices* dialog.
11. Select “Specified spacing” from the *Specify* drop-down.
12. Enter “20.0” as the *Average spacing*.
13. Click **OK** to close the *Redistribute Vertices* dialog.
14. Deselect the channel bank arcs by clicking anywhere other than an arc.

2.3 Creating the Polygons

Polygons must be created from the feature arcs in order to build a mesh by doing the following:

1. Select *Feature Objects* | **Build Polygons**.
2. Using the **Select Feature Polygon** tool, double-click inside the channel to bring up the *2D Mesh Polygon Properties* dialog.
3. In the *Mesh Type* section, select “Patch” from the drop-down.
4. Click the **Preview Mesh** button to bring up an error message about overlapping elements.
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:

7. With the **Create Feature Arc** tool, create sixteen perpendicular arcs across the channel as shown in Figure 5.
8. Using the **Select Feature Arc** tool while holding down the *Shift* key, select all sixteen of the section arcs.
9. Right-click and select **Redistribute Vertices…** to bring up the *Redistribute Vertices* dialog.
10. Select “Number of segments” from the *Specify* drop-down.
11. Enter “10” as the *Number of segment*.
12. Click **OK** to close up the *Redistribute Vertices* dialog.

With new arcs created, polygons must again be created:

13. Deselect the segment arcs by clicking anywhere other than an arc.

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

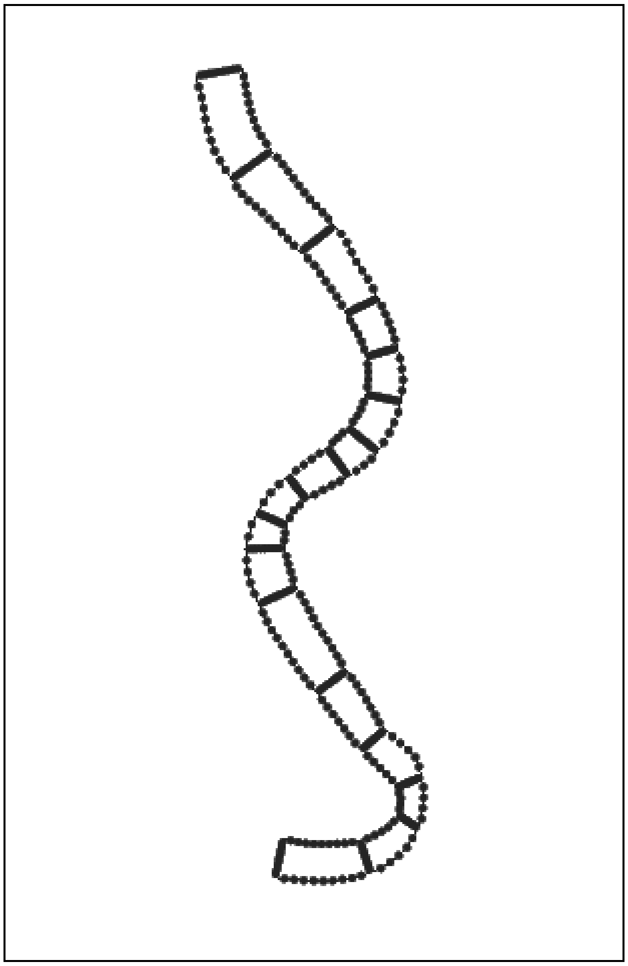


Figure 5 Arcs created across the channel

2.4 Removing Triangular Elements

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

1. Using the **Select Feature Polygon** tool, double-click the southern-most polygon to bring up the *2D Mesh Polygon Properties* dialog.
2. In the *Mesh Type* section, select “Patch” from the drop-down.
3. Click **Preview Mesh**.

Notice that the section of the dialog showing the preview has a number of tools below the preview image. These are the tools referenced in this section.

4. To remove the triangular elements, do the following:
5. Using the **Select Feature Arc** tool while holding the *Shift* key, select both channel bank arcs (in this case, the top and bottom arcs).

6. In the *Arc Options* section, select *Distribute* and accept the suggested number of vertices (or set it to any other reasonable number).
7. Click **Preview Mesh**.

Notice that both channel bank arcs now have the same amount of vertices, and there are no triangular elements.

8. Click **OK** to close the *2D Mesh Polygon Properties* dialog.
9. Repeat steps 1-8 for each of the remaining polygons in order to eliminate all triangular elements. Carefully note the location and orientation of the channel bank arcs for each segment as they will not always be the top and bottom arcs.

2.5 Setting Elevation

An elevation data source must now be specified for each polygon by doing the following:

1. Using the **Select Feature Polygon** tool while holding down *Shift*, select all seventeen polygons.
2. Right-click and select **Attributes…** to bring up the *2D Mesh Multiple Polygon Properties* dialog.
3. Turn on *Mesh type* and select “Patch” from the drop-down.
4. Turn on *Bathymetry type*, and select “Scatter Set” from the drop-down.
5. Click **Scatter Options…** to bring up the *Interpolation* dialog.
6. In the *Interpolation Options* section, select “Single Value” from the *Extrapolation* drop-down.
7. Enter “2.0” as the *Single Value*.
8. Select “Elevation” in the *Scatter Set To Interpolate From* section.
9. Click **OK** to close the *Interpolation* dialog.
10. Click **OK** to close the *2D Mesh Multiple Polygon Properties* dialog.

3 Setting the Boundary Conditions

This model has two boundary conditions: flow and water level.

3.1 Assigning Boundary Conditions

Assign the boundary conditions by doing the following:

1. Using the **Select Feature Arc** tool, double-click the inflow (top) arc in the channel to bring up the *Feature Arc Attributes* dialog.
2. In the *Attribute Type* section, select *Boundary conditions*.
3. Click **Options…** to bring up the *TUFLOW-FV Nodestring Boundary Conditions* dialog.
4. In the tree list on the left, turn on *Water Level*.

5. In the *Water Level* section, in the *Value* column, click **Define…** to bring up the *XY Series Editor* dialog.
6. Outside SMS, browse to the *Data Files* folder for this tutorial and open "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.

The plot in the dialog should appear similar to Figure 6.

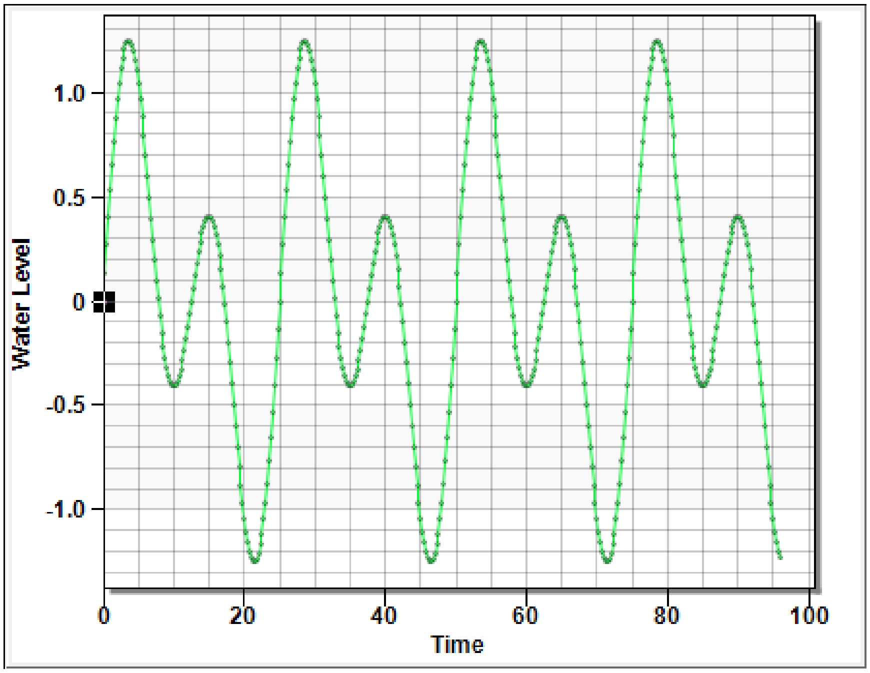


Figure 6 The plot as it should appear once the tide values are entered

9. Click **OK** to close the *XY Series Editor* dialog.
10. Click **OK** to close the *TUFLOW-FV Nodestring Boundary Conditions* dialog.
11. Click **OK** to close the *Feature Arc Attributes* dialog.
12. Repeat steps 1–11, turning on *Flow* in step 4, and using the values from "flow.csv" in steps 6–8.

After repeating step 8, the plot in the dialog should appear similar to Figure 7.

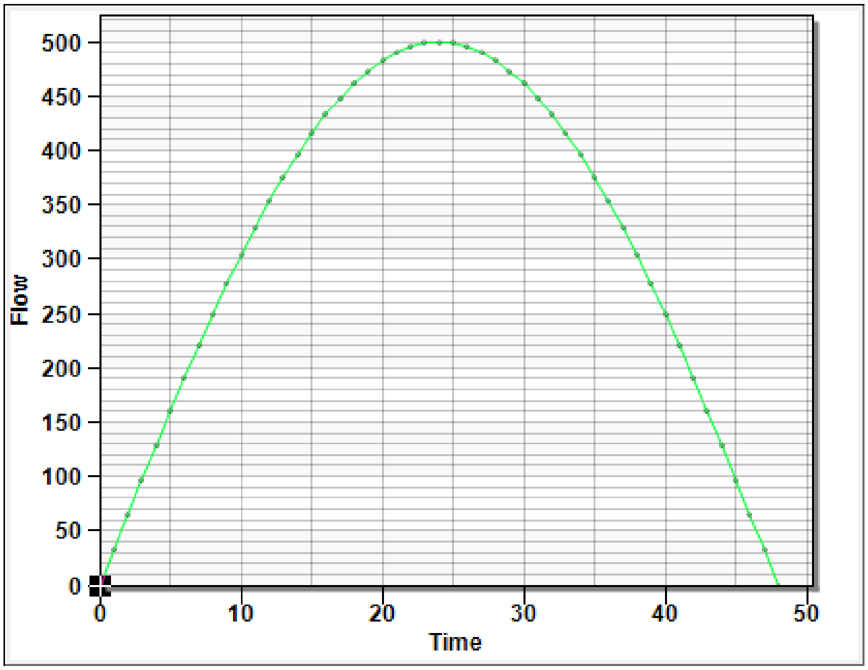


Figure 7 The plot as it should appear once the flow values are entered

3.2 Building a Mesh from Map Data

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* and select “Land Use” from the drop-down.
3. Click **OK** to create the mesh and exit the *2D Mesh Options* dialog.
4. When advised of how many elevations were extrapolated, click **OK** to bring up the *Mesh Name* dialog.
5. Accept the default *Mesh name* and click **OK** to close the *Mesh Name* dialog.
6. Click **Display Options** to bring up the *Display Options* dialog.
7. Select “2D Mesh” from the list on the left.
8. On the *2D Mesh* tab, turn on *Elements*, *Contours*, and *Nodestrings*.
9. On the *Contours* tab, in the *Contour method* section, select “Color Fill” from the first drop-down.
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.

The project should appear similar to Figure 8.

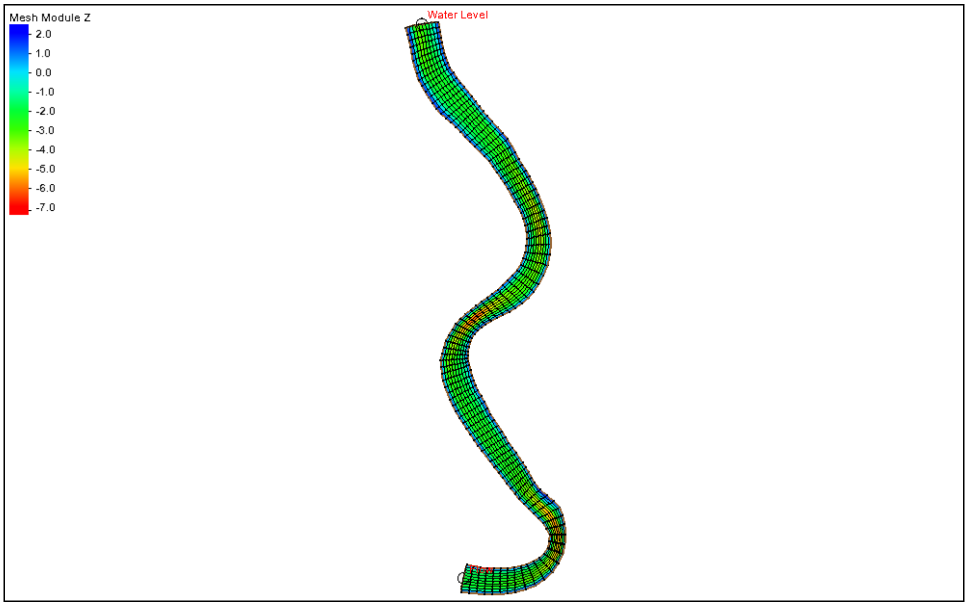


Figure 8 The channel with only mesh data visible

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. On the *Time* tab, select "Hours" from the drop-down in the *Value* column on the *Time Format* row.
4. Enter "0" in the *Value* column on the *Start Time* row.
5. Enter "48" in the *Value* column on the *End Time* row.
6. On the *Output* tab, check the box in the *Value* column on the *SMS Dat Output* row.
7. Enter "h,v" in the *Value* column on the *Dat Output Types* row.
8. Enter "900" in the *Value* column on the *Dat Output Interval* row.

For this tutorial, the settings on the *General*, *HD Parameters*, and *Advanced Commands* tabs should be left unchanged.

9. 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. Select "Sand" from the list on the left.
3. On the *General* tab, enter "0.035" in the *Value* column in the spreadsheet.
4. Repeat steps 2–3, selecting "Gravel" in step 2 and entering "0.028" in step 3.
5. Repeat steps 2–3, selecting "Vegetation" in step 2 and entering "0.06" in step 3.
6. Click **OK** when done to close the *TUFLOW-FV Material Properties* dialog.

6 Saving the Project

Before running the model, save the project:

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 "TUFLOW_FV.sms" as the *File name*.
4. Click **Save** to save the project under the new name and close the *Save As* dialog.

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".

7.1 Copying Required Files

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

1. In Windows Explorer, browse to the *models\TUFLOWFV* (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 *Data Files* directory for the tutorial and paste the files (*Ctrl-V*).
4. In SMS, select *Edit* | **Preferences…** to bring up the *SMS Preferences* dialog.
5. On the *File Locations* tab, in the *Model Executables* section, scroll down to the "Generic" model, and click on the button in the *Executable* column to bring up the *Select model executable* dialog. The button name will be either **BROWSE** or it will be the path to the BAT file.
6. Select "All Files (*.*)" from the *Files of type* drop-down.
7. Browse to the tutorial directory and select "convert_and_run.bat".
8. Click **Open** to exit the *Select model executable* dialog.
9. Click **OK** to close the *SMS Preferences* dialog.

7.2 Editing the Batch File

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

1. Outside of SMS, open “convert_and_run.bat” into a text editor.
2. Edit the line starting with “set parser=” so it points to the tutorial directory containing “mesh_to_FV.exe” (Figure 9).
3. Edit the line starting with “set tf_fv=” so it points to the tutorial directory containing “TUFLOWFV.exe”.
4. Once edited, save the BAT file and close it.

```
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 9 The "set parser" and "set tf_fv" lines need to be edited

7.3 Running TUFLOW FV

1. Insert the TUFLOW-FV dongle into a free USB port on the computer.
2. In SMS, select *TUFLOW-FV* | **Run TUFLOW-FV**.
3. When advised that no model checks were violated, click **OK**.
4. 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.

5. Click **Open** and browse to the *Data Files\TUFLOWFV\output* directory.
6. While pressing the Shift key, select both “TUFLOW_FV_H.dat” and “TUFLOW_FV_V.dat”.
7. Click Open to import the DAT files and exit the Open diarectory.

This adds 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. Click **Display Options** to bring up the *Display Options* dialog.
2. Select "2D Mesh" from the list on the left.
3. On the *2D Mesh* tab, click **All Off** and turn on *Contours*, *Elements* and *Vectors*.
4. On the *Contours* tab, in the *Contour method* section, select "Color Fill" from the first drop-down.
5. Click **OK** to exit the *Display Options* dialog.
6. Click through the new datasets and time steps.

The upstream end of the mesh is shown in Figure 10 at time step "1 04:15:00" with the " TUFLOW_FV_V" vector and " TUFLOW_FV_H" scalar datasets active.

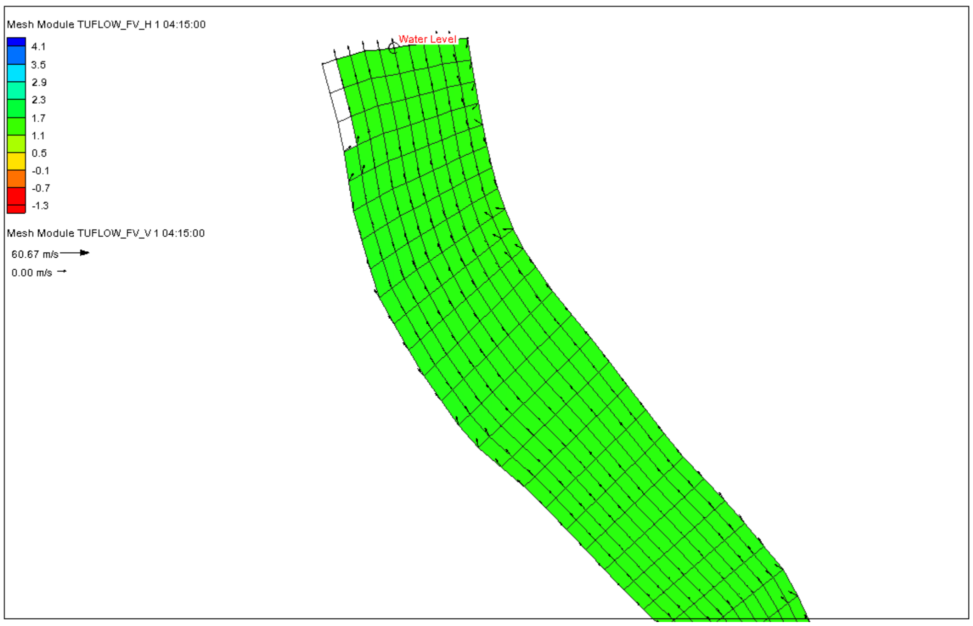


Figure 10 A portion of the mesh with TUFLOW_FV_V and TUFLOW_FV_H active

9 Conclusion

This concludes the "TUFLOW FV" tutorial. Feel free to continue experimenting, or exit SMS.