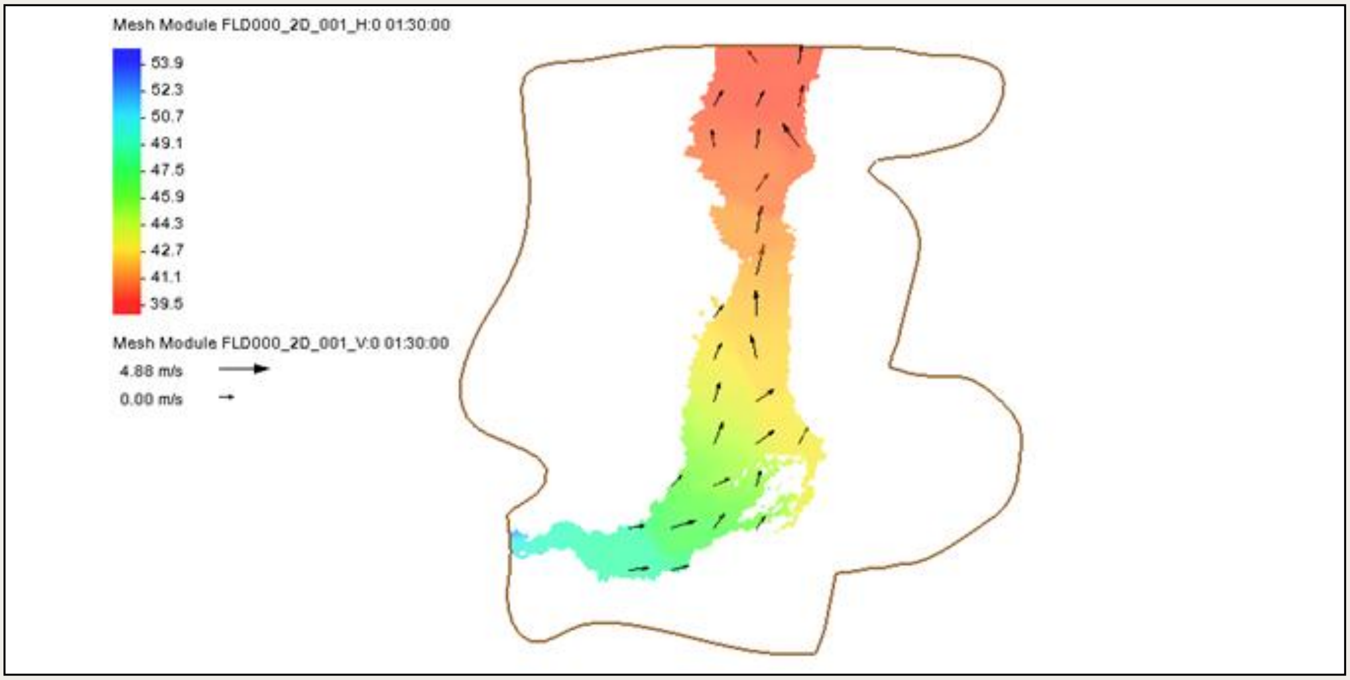
SMS 14.0 Tutorial

TUFLOW FV

Set up a TUFLOW FV model



Objectives

This tutorial demonstrates creating a simple model of a short section of river using the SMS TUFLOW FV interface.

Prerequisite Tutorials

- Overview

Required Components

- SMS Core
- TUFLOW FV Model

Time

- 20–30 minutes

AQUAVEO™

1 Introduction

TUFLOW FV is an engine for performing 2D and 3D hydrodynamic simulations. It has been applied extensively to both coastal and riverine conditions. The model solves the Non-Linear Shallow Water Equations (NLSWE) on a flexible mesh using a finite-volume numerical scheme.

The TUFLOW FV manual contains references and descriptions of the details of the model features. The TUFLOW FV user manual can be found at:

https://downloads.tuflow.com/_archive/TUFLOW_FV/Manual/TUFLOW_FV_User_Manual_2020.pdf

This tutorial uses a fictional site that is coded in the engine to work without a license for demonstration purposes.

An initial project has been provided for this exercise that includes the mesh of the domain. For information on constructing a mesh of this type, refer to one of the tutorials on mesh generation.

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

1. Select the *File* | **Open…** menu command to bring up the *Open* dialog.
2. Browse to the *data_files* directory for this tutorial and select "intro.sms".
3. Click the **Open** button to exit the *Open* dialog and import the project.

This project includes:

- A mesh named " Mesh_w_Roads".
- A material coverage that contains a single polygon bounding the mesh with a single material type named " Base material". This tutorial explains how to build materials and properties into a coverage.
- A framework of map coverage folders. Since a project often contains multiple coverages, the best practice is to organize these into folders. The approach demonstrated here is one option to illustrate this type of organization.

In the Graphics Window, the project should appear similar to Figure 1.

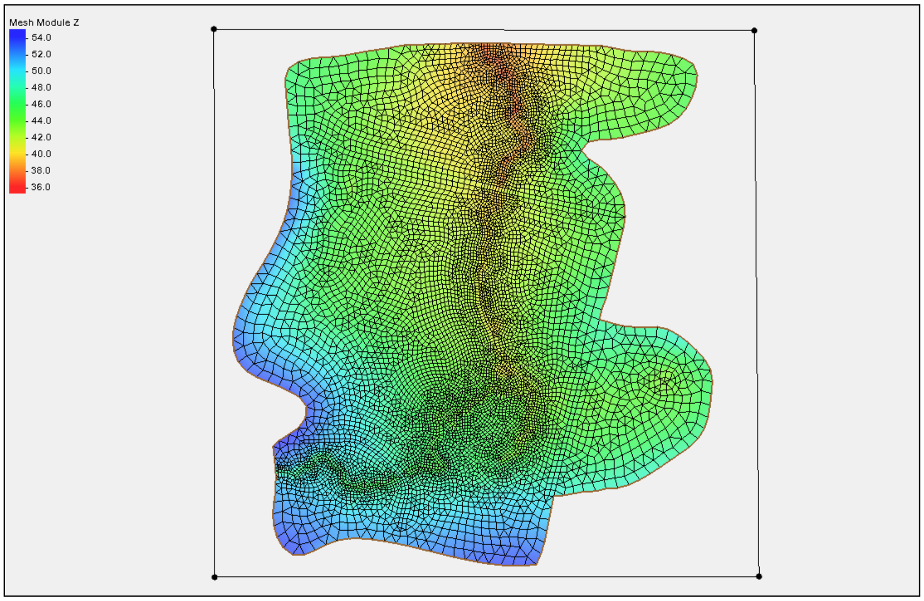


Figure 1 Initial project

2 Creating a Simulation

The TUFLOW FV interface is designed to use a simulation-based modeling approach. This means that one or more simulation objects that link together building blocks or components of a site analysis will be created. A simulation includes:

- A simulation object with specified parameters.
- An unstructured mesh which is the basis of the numerical calculations.
- One or more material definition layers.
- One or more boundary condition layers.

To create a simulation object:

1. In the Project Explorer below the tree (in the white space), right-click and select the *New Simulation* | **TUFLOWFV** context menu command.

A new simulation object named “ Sim” will appear in the Project Explorer.

2. Right-click on the new “ Sim” simulation object and select the **Rename** context menu command.
3. For the name for the simulation, enter “FLD000_2D_001” and press the *Enter* key to set the new name.

A typical project will include several simulations to experiment with design alternatives, flow conditions and other variations.

3 Setting the Material Properties

The TUFLOW FV engine supports a multi-layered approach to specifying the spatially variable resistance to flow and other material attributes. The bottom level of this approach is a material coverage that should cover the entire domain. This coverage can include multiple material polygons if desired. In fact, it can represent the entire classification of material zones. The materials are specified in one or more material coverages which are mapped in the order they appear on the tree onto the mesh/grid. Normally, the base coverage will include polygons that cover the entire domain.

To illustrate this, this example will use two layers.

3.1 The Base Material

For this exercise the coverage " Base material" was provided with one polygon that covers the domain. Note: As illustrated it does not need to match the domain.

To assign the base material:

1. Select the " Base material" map data coverage to make it active.
2. Right click on the " Base material" coverage and change the coverage type to "*Models* | *TUFLOWFV* | **Materials**".
3. Select the **Select Polygon** dynamic tool and double-click inside the polygon to open the *Assign Material* dialog.

The dialog includes a list of all the material types used by this coverage. New types can be created as needed.

4. Click on the **Add Material** button to create a new material.
5. Change the *Name* of the new material from "new material" to "Base".
6. In the "Base" row, turn on the *Override Bottom Roughness* checkbox.
7. Set the *Bottom Roughness* to "0.04" to define the default roughness.
8. Click the **OK** button to close the *Assign Material* dialog.

All cells/elements inside the polygon will now be defaulted to the "Base" material type with a roughness of 0.04.

3.2 Material from a Shapefile

The second material layer used by this simulation will come from a shapefile. To assign this material layer:

1. Select the *File* | **Open…** menu command to bring up the *Open* dialog.
2. Browse to the *data_files/gis* directory for this tutorial and select the "2d_mat_M01_002_R.shp" shapefile.
3. Click the **Open** button to exit the *Open* dialog and import the shapefile.
4. Right-click on the " Map Data" folder and select the **New Coverage** context menu command to bring up the *New Coverage* dialog.
5. For the *Coverage Type*, in the *Models* section under *TUFLOWFV,* select *Materials*.
6. For the *Coverage Name,* enter "2d_mat_M01_002_R".
7. Click the **OK** button to close the *New Coverage* dialog.
8. Drag the new " 2d_mat_M01_002_R" map coverage into the " Materials" folder.
9. In the *GIS Data* folder, right-click on the " 2d_mat_M01_002_R.shp" shapefile and select the *Convert* > **Shapes → Feature Objects** context menu command to open the *GIS to Feature Objects Wizard* dialog.

10. Confirm that the *Select a coverage for mapping* option is set to "2d_mat_M01_002_R" and click the **Next >** button.
11. Accept the default setting and click the **Finish** button close the *GIS to Feature Objects Wizard* and convert the polygons in this shapefile into polygons on the selected map coverage.

Assigning Material Properties to Polygons

Now, to assign a roughness for these polygons by doing the following:

1. Select the " 2d_mat_M01_002_R" map coverage to make it active.
2. Using the **Select Polygon** dynamic tool, right-click in the Graphics Window and select the **Select all…** context menu command.
3. With all the polygons selected, right-click in the Graphics Window and select the **Assign Material** context menu command to bring up the *Assign Materials* dialog.
4. Click on the **Add Material** button to create a new material type.
5. In the second *Name* column, enter "Low roughness".
6. In the "Low roughness" row, turn on the *Override Bottom Roughness* checkbox.
7. For the *Bottom Roughness*, enter "0.02".
8. In the *Global export format* section, set the *TUFLOWFV simulation export file format* drop-down option to "Shapefile".

This instructs the SMS interface to write this layer of the material definition as a shapefile. This option allows the separation of this material specification from the other 2DM specification. If the option is left as a 2dm, the two material layers would be combined when exported to the 2DM file.

9. Click on the number *2* to select the entire row for material assignment.

This tells SMS to apply the material assignment to that row.

10. Click the **OK** button to close the *Assign Materials* dialog.

This has assigned a lower roughness value to the polygons that were defined in the shapefile. This lower value will be mapped to cells that lie inside these polygons.

4 Setting the Boundary Conditions

Boundary conditions for TUFLOW FV can also be defined in multiple layers. For this tutorial single layer is used.

4.1 Creating Boundary Condition Arcs

To define boundary conditions, first the place where the boundary conditions will be applied in the model domain must be defined. To do this:

1. Right-click on the " Map Data" folder and select the **New Coverage** context menu command to bring up the *New Coverage* dialog.
2. For the *Coverage Type*, in the *Models* section under *TUFLOWFV*, select *Boundary Conditions*.
3. For the *Coverage Name*, enter "Open_BCs_001".

4. Click the **OK** button to close the *New Coverage* dialog.
5. Drag the new " Open_BCs_001" coverage into the " BCs" map data folder.
6. Make sure the " Mesh_w_Roads" mesh is turned on in the Project Explorer to provide spatial reference.
7. Select the " Open_BCs_001" coverage to make it active.
8. Use the **Create Feature Arc** dynamic tool to digitize the outflow (upper) and inflow (lower left) locations of this simulation as shown in Figure 2.

These are external boundaries, so it is suggested that these arcs be just outside of the mesh as shown.

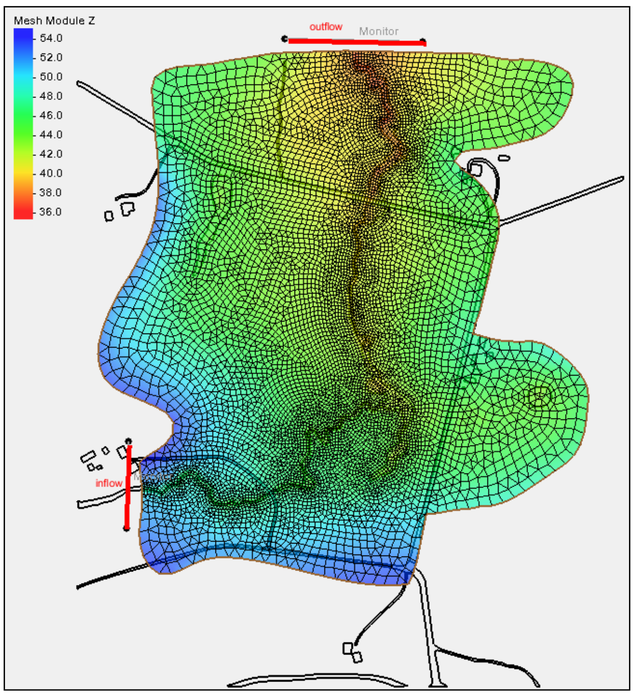


Figure 2 Position of the BC arcs

4.2 Assigning Boundary Conditions

Assign the boundary conditions by doing the following:

Defining Lower Arc

1. Using the **Select Feature Arc** dynamic tool, double-click the inflow (lower left) arc to bring up the *Assign Boundary Condition* dialog.
2. In the *BC options* tab, in the *Options* section, from the *Type* drop-down menu, select "Q (Nodestring Flow)".
3. From the *Subtype* drop-down menu, select "Sub-type 4".

Note: This subtype specifies the distribution of the flow along the arc. Refer to the TUFLOW FV user manual for details.

4. Click the **Define Curve…** button to bring up the *BC Curve Editor* dialog.
5. Change the *Number of rows* to "1" to create a row.

6. Outside SMS, browse to the *data_files* folder for this tutorial and open "flow.csv" in either a text or spreadsheet editor.
7. *Copy* (*CTRL+C*) all values under the *X* and *Y* columns (A2 through B42).
8. Back in the *BC Curve Editor* dialog, select the first cell under and *paste* (*CTRL+V*) the values from the CSV file.

The hydrograph defined in the flow file will appear as shown in Figure 3.

9. Click the **OK** button to close the *BC Curve Editor* dialog.
10. Click the **OK** button to close the *Assign Boundary Condition* dialog.

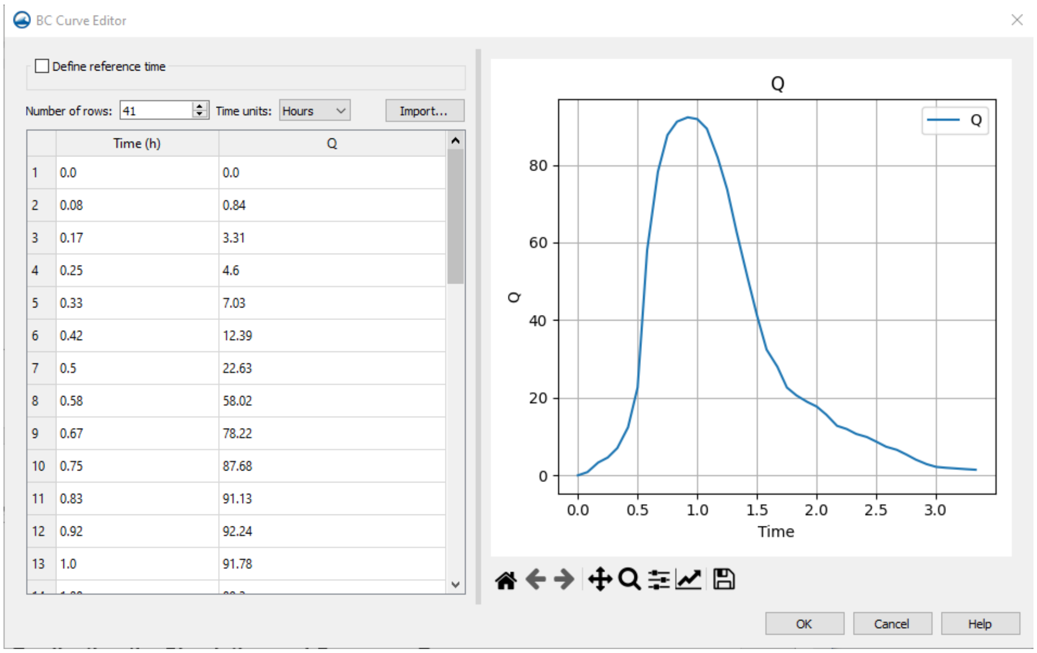


Figure 3 The inflow boundary condition time series curve

Defining Upper Arc

1. Using the **Select Feature Arc** dynamic tool, double-click the outflow (upper) arc to bring up the *Feature Arc Attributes* dialog.
2. In the *BC options* tab, in the *Options* section, from the *Type* drop-down menu, select "WL (Water Level)".
3. From the *Subtype* drop-down menu, select "Sub-type 1".

Note: This subtype specifies the velocities will be computed by TUFLOW FV. Refer to the TUFLOW FV user manual for details.

4. Click the **Define Curve…** button to bring up the *BC Curves Editor* dialog.
5. Change the *Number of rows* to "2" to create two rows.
6. In row *1*, enter "0.0" for the *Time* and "40.0" as the *WL*.
7. In row *2*, enter "3.33" for the *Time* and "40.0" as the *WL*.

This defines a constant outflow water level.

8. Click the **OK** button to close the *BC Curves Editor* dialog.
9. In the *Assign Boundary Condition* dialog, turn on the *Define default* checkbox.

10. For the *WL default* value, enter “40.0”.
11. Turn on the *Define offset* checkbox.
12. For the *WL offset* value, enter “0.0”.
13. Turn on the *Define scale* checkbox.
14. For the *WL scale* value, enter “1.0”.
15. Click the **OK** button to close the *Assign Boundary Condition* dialog.

The boundary conditions are now defined.

5 Linking the Model Components

Now the defined components for this simulation need to be added to the simulation. To do this:

1. Drag the “ Mesh_w_Roads” mesh into the “ FL000_2D_001” simulation.

When dragging, a line will appear below the simulation when the cursor is in the appropriate location to drop this mesh into the simulation.

2. Right-click on the “ Base material” map coverage and select the *Apply to…* | **TUFLOWFV Simulations > FLD000_2D_001** context menu command.
3. Right-click on the “ *2d_mat_M01_002_R*” map coverage and select the *Apply to…* | **TUFLOWFV Simulations > FLD000_2D_001** context menu command.

Note: the link to this coverage appears below the link to the “ Base material” coverage. It is important that the base material is assigned first and then the shapefile material to be applied later in order to override the base.

4. Right-click on the “ Open_BCs_001” map coverage and select the *Apply to…* | **TUFLOWFV Simulations > FLD000_2D_001** context menu command.

6 Setting the Model Parameters

Next, assign the model parameters by doing the following:

1. Right-click on the “ FL000_2D_001” simulation and select the **Model Control…** context menu command to bring up the *TUFLOWFV Model Control* dialog.

General Tab

1. On the *General* tab, turn on the *Define spatial order* checkbox.
2. From the *Horizontal* drop-down menu, select “1st”.
3. Turn on the *Define hardware solver* checkbox.
4. From the drop-down menu, select “CPU”.
5. Turn on both the *Warn on projection mismatch of GIS inputs* and *Tutorial model* checkboxes.

Time Tab

1. On the *Time* tab, for the *Starting time,* enter “0.0”.
2. For the *Ending time,* enter “3.0”.

Global Parameters Tab

1. On the *Global parameters* tab, turn on the *Define bottom drag model* checkbox.
2. From the *Method* drop-down, select *Manning.*
3. For the *Manning's n coefficient,* enter "0.06".
4. From the *Horizontal mixing model* drop-down, select "Smagorinsky".
5. For the *Global horizontal eddy viscosity coefficient*, enter "0.4".
6. Turn on the *Define global horizontal eddy viscosity limits* checkbox.
7. For the *Minimum eddy viscosity*, enter "0.05".
8. Turn on the *Define stability limits* checkbox and leave the values at the defaults.

Wind Stress Tab

1. On the *Wind stress* tab, turn on the *Define wind model* checkbox.
2. From the *Model* drop-down, select "Wu".

Geometry Tab

1. On the *Geometry* tab, turn on the *Define cell wet/dry depths* checkbox.
2. For the *Dry depth,* enter "0.001".
3. For the *Wet depth,* enter "0.02".

Initial Conditions Tab

1. On the *Initial conditions* tab, turn on the *Define initial water level* checkbox.
2. For the *Define initial water level* value, enter "40.0".

Output Tab

1. On the *Output* tab, in the *Output blocks* section, click the **Add Row** button three times to create three rows.

Row 1

1. In the first row of the *Output blocks* table, select "DATV".

This defines the nodal datasets that will be created by the engine. The NetCDF option creates cell centered datasets and requires a UGrid geometry.

2. With the first row selected, in the *Output block options – Row 1* section set:
3. Turn on the *Define interval* checkbox.
4. For the *Output interval (s),* enter "600.0".
5. Leave other values at default.

Row 2

1. In the second row of the *Output blocks* table, from the drop-down, select *Flux.*
2. With this row selected, in the *Output block options – Row 2* section set:
3. Turn on the *Define interval* checkbox.
4. For the *Output interval (s),* enter "900.0".

5. Leave other values at default.

Row 3

1. In the third row of the *Output blocks* table, from the drop-down, select *Mass.*
2. With this row selected, in the *Output block options – Row 3* section set:
3. Turn on the *Define interval* checkbox.
4. For the *Output interval (s),* enter “900.0”.
5. Leave other values at default.

Directories section

1. In the *Directories* section, enter the following:
2. For the *Log directory,* enter “log”.
3. For the *Output directory,* enter “..\results”.
4. Turn on the *Write restart file* checkbox.
5. For the *Write restart dt (hours)*, enter “1.0”.
6. Turn on the *Enable restart overwrite* checkbox.
7. Turn on the *Write check files* checkbox.
8. For the *Check files directory,* enter “..\check”.

Materials Tab

1. On the *Materials* tab, turn on the *Define default material (set mat)* checkbox.
2. Turn on the *Override Bottom Roughness* checkbox.
3. For the *Bottom Roughness* value, enter “0.04”.

Boundary Conditions Tab

1. On the *Boundary conditions* tab, turn on the *Define BC default update dt* checkbox.
2. For the *BC default update dt (s)* value, enter “300.0”.
3. Click **OK** to close the *TUFLOWFV Model Control* dialog.

7 Saving the Project

Before running the model, save the project:

1. Select the *File* | **Save As...** menu command to bring up the *Save As* dialog.
2. For the *File name,* enter “Intro_done.sms”.
3. Click the **Save** button to save the project under the new name and close the *Save As* dialog.

8 Running the Model

This tutorial uses a specific location that TUFLOWFV recognizes as a “demo” or “tutorial” location. Therefore, no license for the engine is required. This was specified in the *TUFLOWFV Model Control* dialog.

To run the model, do the following:

1. Right-click on the " FL000_2D_001" simulation and select the **Save Simulation** context menu command.
2. SMS may prompt with a warning that coverages will be renumbered. If it does, click the **Yes** button to continue.
3. Once the command is complete, open a file browser and go to the directory where the project files were saved.
4. In the operating system, review the "intro_done_models" folder.

This folder was created in a typical file structure used by the TUFLOW FV.

5. In SMS, in the Project Explorer, right-click on the " FL000_2D_001" simulation and select the **Run simulation** context menu command.

The *Simulation Run Que* dialog will appear and monitor the progress of TUFLOW FV. After a couple of minutes (depending on machine speed), a message indicating a successful simulation run should appear.

6. Click the **Load Solution** button to import the solution data.
7. Close the run queue dialog by clicking the **Close** button.

Based on the setting in the *Output* tab of the model control, TUFLOW FV created the following mesh datasets which were loaded as part of the solution:

- " FLD000_2D_001_D" – transient water depth
- " FLD000_2D_001_H" – transient water surface elevation
- " FLD000_2D_001_V" – transient water current (vector function)

Additional datasets can be enabled in the output tab of the model control dialog. Review the model results using the tools illustrated in the data visualization tutorial.

8. From the *Solution* simulation folder, select the " FLD000_2D_001_D" dataset to make it active.
9. In the *Time Step* window, select the last time step. The *Graphics Window* will appear similar to Figure 4.

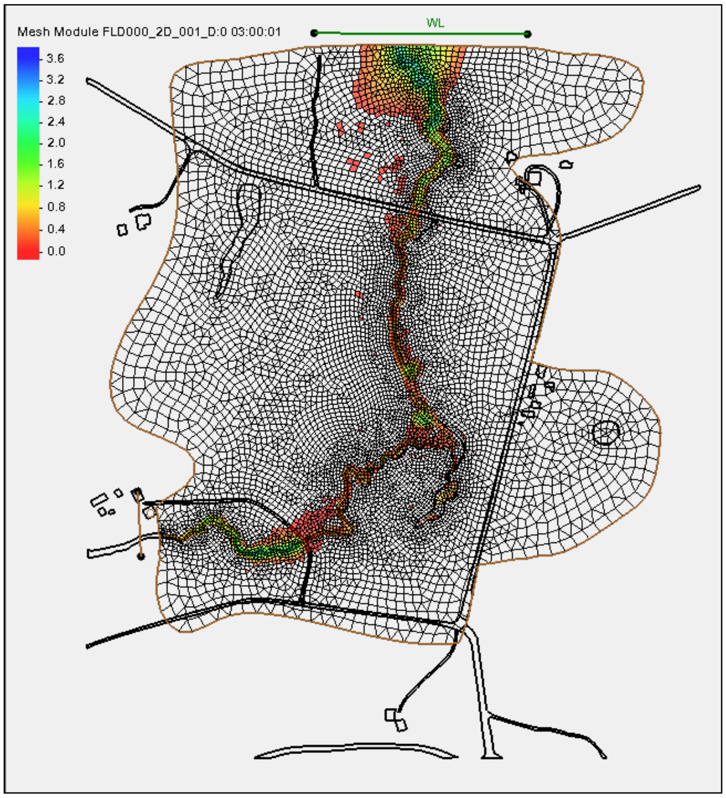


Figure 4 The last time-step for the water depth solution

9 Conclusion

This concludes the introductory “TUFLOW FV” tutorial. Additional tutorials will illustrate many other features of this versatile numeric engine.