RiverFlow2D
Two-Dimensional
River Dynamics Model

TUTORIALS

December, 2017

Hydronia LLC

RiverFlow2D© model and documentation produced by Hydronia, LLC, Pembroke Pines, FL. USA.
Information in this document is subject to change without notice and does not represent a commitment on part of Hydronia, LLC. The software described in this document is furnished under a license agreement.
RiverFlow2D, OilFlow2D, RiverFlow2D, and RiverFlow2D GPU are copyrighted by Hydronia, LLC. 2011-2017.
SMS is trademark of Aquaveo, LLC.
All other products or service names mentioned herein are trademarks of their respective owners.
All rights reserved. No part of this publication may be reproduced, stored in a retrieval system or transmitted in any form or by any means electronic, mechanical, photocopying, recording or otherwise, without the prior written permission of Hydronia, LLC.
Last document modification date: September, 2017.
Technical Support: `support@hydronia.com`

Web site: `www.hydronia.com`

Contents

List of Figures v
List of Tables ix
1 Introduction 1
2 Creating your First RiverFlow2D Project 3
2.1 Run SMS and Open RiverFlow2D Template 4
2.2 Define Projections and Units 4
2.3 Read in Initial Data 5
2.3.1 Read in Aerial Image 5
2.3.2 Read in Topographic Data 5
2.3.3 Display Options 5
2.4 Define Model Domain 6
2.4.1 Define Coverage Type 6
2.4.2 Create a Polygon to Define the Model Extent 7
2.4.3 Define Arc Spacing 8
2.4.4 Redistribute Arcs Vertices 8
2.4.5 Define Polygon Attributes 9
2.5 Setting RiverFlow2D Boundary Conditions 10
2.5.1 Feature Arc Attributes 10
2.6 Generate the Mesh 12
2.7 Assign Manning's n to Materials 14
2.8 Assign RiverFlow2D Parameters 15
2.9 Running RiverFlow2D 15
3 Visualizing RiverFlow2D Results 17
3.1 Scalar Dataset Options 17
3.1.1 Vector Dataset Options 18
4 Simulating Bridges 21
4.1 Create a Bridge Geometry File 21
4.2 Open an Existing Project 23
4.3 Enter the Bridge Polyline in the Default Coverage 23
4.4 Generate the Mesh 25
4.5 Running the Model 26
5 Simulating Culverts 27
5.1 Open an Existing RiverFlow2D Project 28
5.1.1 Create the Culvert 29
5.2 Setup Control Parameters 31
5.3 Running the Model 32
5.4 Review Culvert Output Files 33
6 Simulating Levees using Weirs 35
6.1 Open an Existing Project 35
6.2 Import a Weir Breakline 36
6.3 Convert the Weir Breakline to the Map Coverage 38

6.4 Generate the Mesh . . . 40
6.5 Running the Model . . . 41
7 Hydrologic Simulations . . . 43
7.1 Open an Existing Project . . . 44
7.2 Create New Materials Corresponding to Rainfall and Infiltration Files . . . 44
7.3 Create the Rainfall and Evaporation Time-series Data File . . . 45
7.4 Create the Infiltration Parameters Data File . . . 47
7.5 Create the Rainfall/Evaporation Polygons . . . 47
7.6 Create the Infiltration Polygons . . . 49
7.7 Generate the Mesh and Copy Rainfall and Infiltration Files to the Mesh Folder . . . 51
7.8 Running the Model . . . 51
7.9 Visualizing Rainfall/Infiltration Results . . . 53
8 Setting up Initial Water Surface Elevations . . . 55
8.1 Open an Existing Project . . . 55
8.2 Create New Initial Water Surface Elevation Value . . . 56
8.3 Create the Initial Water Elevation Polygon . . . 57
8.4 Generate the Mesh . . . 59
8.5 Running the Model . . . 59
9 Using Manning's n Shapefiles . . . 61
9.1 Open an Existing RiverFlow2D Project . . . 61
9.2 Create a New Area Property Coverage for Manning's n . . . 62
9.3 Read in the Shapefiles with Manning's n Polygons . . . 62
9.4 Map the Shapefile to the Manning's n Coverage . . . 63
9.5 Map the Manning's n Coverage to the Existing Mesh . . . 64
9.6 Display the Manning's n on the Existing Mesh . . . 65
9.7 Enter n-values for Each Material . . . 66
9.8 Save the Project and Check that Manning's n have been Updated in the .2dm file . . . 67
10 Simulating Bed Load Sediment Transport with Limited Erosion Bed Areas . . . 69
10.1 Open an Existing Project . . . 69
10.2 Create New Maximum Erosion Depth Values . . . 70
10.3 Create the Maximum Erosion Depth Polygon . . . 71
10.4 Generate the Mesh . . . 73
10.5 Set the Global Parameters . . . 73
10.6 Running the Model . . . 74
10.7 Post-processing the Model Results . . . 75
11 Suspended Sediment Transport Simulations . . . 77
11.1 Run SMS and Open the Existing Project File . . . 77
11.2 Set suspended sediment data in the RiverFlow2D Data Input Program . . . 78
11.3 Add the Inflow Concentrations for each Fraction . . . 79
11.4 Run RiverFlow2D from the Data Input Program . . . 81
11.5 View RiverFlow2D Sediment Transport Model Results . . . 82
12 Wind Driven Circulation . . . 85
12.1 Open an Existing Project . . . 86
12.2 Create New Materials Corresponding to Wind Velocity Files . . . 86
12.3 Create the Wind Velocity Time-series Data File . . . 87
12.4 Create the Wind Polygons . . . 88
12.5 Generate the Mesh and Copy Wind Velocity Files to the Mesh Folder . . . 89
12.6 Running the Model . . . 90
12.7 Visualizing Wind Driven Circulation Results . . . 92
13 Simulating an Internal Dam Breach . . . 95
13.1 Create a Breach Time Series Data File . . . 95

13.2 Enter the Dam Polyline in the Default Map Coverage . . . 97
13.3 Enter the initial water surface elevations . . . 99
13.4 Generate the Mesh . . . 101
13.5 Running the Model . . . 102
14 Post-processing Calculations . . . 105
14.1 Creating an Observation Coverage . . . 105
14.2 Cross Sections . . . 106
14.3 Profiles . . . 108
15 Interpolation Methods . . . 111
16 Using Huge Elevation Data Sets . . . 113
17 Converting .DXF Files to Scatter Data . . . 115
18 Using HEC-RAS Data Extraction Tool . . . 117
18.1 Read HEC-RAS Geometry File and Create the X Y Z and Polygon Files . . . 117
19 Investigating the Mesh Quality . . . 121
20 Refining Elements . . . 123
20.1 Inserting Breaklines . . . 123
20.2 Arc Inserted on the Conceptual Model Coverage . . . 124
21 Creating New Data Sets Using the Data Calculator . . . 127
22 Exporting Shapefiles . . . 129

List of Figures

2.1 Elevation data and background image. . . . 6
2.2 Upstream boundary. . . . 7
2.3 Downstream boundary. . . . 7
2.4 Model extent arcs defined. . . . 8
2.5 Final Conceptual Model coverage. . . . 9
2.6 Upstream zoomed area. . . . 11
2.7 Upstream node string. . . . 11
2.8 SMS XY Series Editor used to enter hydrograph boundary condition. . . . 12
2.9 Mesh with elevation contours. . . . 13
2.10 3D view of the mesh. . . . 14
2.11 RiverFlow2D Data Input Program main window. . . . 15
2.12 Model run report window. . . . 16

3.1 Shaded contour plot for the Water Surface Elevation dataset. . . . 18
3.2 Velocity vectors. . . . 19
3.3 Flow trace animation. . . . 20

4.1 Bridge front view. . . . 22
4.2 Bridges panel in the RiverFlow2D Data Input Program. . . . 23
4.3 Existing RiverFlow2D project. . . . 24
4.4 Bridge polyline in the Domain Outline layer. . . . 24
4.5 Feature arc attributes dialog. . . . 25
4.6 Boundary conditions dialog. . . . 25
4.7 Mesh aligned with bridge polyline. . . . 26

5.1 Culvert scheme. Colors indicate the bed elevations where blue are the lowest areas and red the highest. . . . 27
5.2 Culvert tutorial. Initial window. . . . 29
5.3 Save data to .2dm file. . . . 31
5.4 Control data. . . . 32
5.5 Control data. . . . 32
5.6 Graphic output. . . . 33
5.7 Output file for Culvert A. . . . 34

6.1 Existing RiverFlow2D project. . . . 36
6.2 Open File Format dialog. . . . 37
6.3 File Import Wizard dialog step 1 of 2. . . . 37
6.4 File Import Wizard dialog step 2 of 2. . . . 37
6.5 Scatter Breakline Options dialog. . . . 38
6.6 Scatter Breaklines to Map conversion command. . . . 38
6.7 Interpolate to Map Command. . . . 39

6.8 Map data showing domain boundary and internal breakline. 39
6.9 Feature Arc Attributes dialog. 40
6.10 Weir Boundary Conditions data. 40
6.11 Mesh aligned with weir polyline. 41

7.1 Existing RiverFlow2D project. 44
7.2 Materials Data Dialog to enter Rainfall/Evaporation and Infiltration data file names. 45
7.3 Rainfall intensity time series. 46
7.4 Rainfall/Evaporation polygon. 47
7.5 Polygons must be independent. 48
7.6 Convert vertices to nodes. 48
7.7 Assign Rainfall/Infiltration data file name to polygon. 48
7.8 Select rainfall data file name for polygon. 49
7.9 Infiltration polygon. 49
7.10 Convert vertices to nodes. 50
7.11 Assign infiltration data file name to polygon. 50
7.12 Select infiltration data file name for polygon. 50
7.13 Second infiltration polygon. 51
7.14 The project folder and the mesh share the same name. 52
7.15 Control Data panel. 52
7.16 RiverFlow2D window at the end of the Rainfall/Infiltration tutorial run. 53
7.17 Visualization of rainfall and infiltration results. 54

8.1 Existing RiverFlow2D project. 56
8.2 Materials Data Dialog to enter initial water elevation. 57
8.3 Initial water elevation polygon. 57
8.4 Convert vertices to nodes. 58
8.5 Polygons must be independent. 58
8.6 Assign a water elevation value to polygon. 58
8.7 Select value for polygon. 59
8.8 The project folder and the mesh share the same name. 59

9.1 RiverFlow2D project. 62
9.2 GIS to Feature Objects Wizard dialog. 64
9.3 Mapping polygons to materials. 64
9.4 Display of materials (Manning's n). 66
9.5 RiverFlow2D Material properties. 67

10.1 Existing RiverFlow2D project. 70
10.2 Materials Data Dialog to enter maximum erosion depths. 71
10.3 Maximum erosion depth polygon. 71
10.4 Polygons must be independent. 72
10.5 Convert vertices to nodes. 72
10.6 Assign maximum erosion depth value to polygon. 73
10.7 Select value for polygon. 73
10.8 The project folder and the mesh share the same name. 73
10.9 Bed Load Sediment Transport tab. 74
10.10 Erosion and Deposition Graph. 76

11.1 RiverFlow2D Data Input Program with Sediment Transport Module activated. . . 78
11.2 RiverFlow2D Data Input Program with Suspended Sediment Data. 79
11.3 RiverFlow2D Data Input Program showing the *Open Boundary Conditions* Panel. 80

11.4 RiverFlow2D Data Input Program showing the *Open Boundary Conditions* Panel with the completed concentration column. . . . 81
11.5 Sediment Transport Model Run Report Window. . . . 82
11.6 Sediment Transport Model run result visualization. . . . 83

12.1 Existing RiverFlow2D project. . . . 86
12.2 Materials Data Dialog to enter wind velocity data file names. . . . 87
12.3 Wind polygon. . . . 88
12.4 Convert vertices to nodes. . . . 89
12.5 Assign wind velocity data file name to polygon. . . . 89
12.6 Select wind velocity data file name for polygon. . . . 89
12.7 The project folder pointing to the mesh coverage. . . . 90
12.8 Control Data panel. . . . 91
12.9 RiverFlow2D window at the end of the Wind Simulation tutorial run. . . . 92
12.10 Visualization of wind simulation results. . . . 93

13.1 Dam Breach Representation in RiverFlow2D. . . . 96
13.2 Dam Breach panel in the RiverFlow2D Data Input Program. . . . 97
13.3 Dam polyline in the main map layer. . . . 98
13.4 Feature arc attributes dialog. . . . 98
13.5 Dam breach boundary conditions dialog. . . . 99
13.6 Polygon used to set the initial water elevations on the reservoir. . . . 99
13.7 Materials Data Dialog to enter initial water elevation. . . . 100
13.8 Convert vertices to nodes. . . . 100
13.9 Assign a water elevation value to polygon. . . . 101
13.10 Select value for polygon. . . . 101
13.11 Mesh aligned with dam polyline. . . . 102
13.12 Dam Breach Component. . . . 102
13.13 Dam Breach Control Parameters. . . . 103
13.14 Dam Breach Run Window. . . . 104

14.1 Cross sections arcs on the Observation coverage. . . . 107
14.2 Profile plot of the geometry of the stream. . . . 108

15.1 Interpolation Control dialog. . . . 112

18.1 Extract data from HEC-RAS command on Tool panel. . . . 118
18.2 Results of HEC-RAS data extraction. . . . 118
18.3 Folder for HEC-RAS extracted files. . . . 119
18.4 Confirmation of HEC-RAS extracted files. . . . 119

19.1 Mesh Quality for Hoh River mesh. . . . 122
19.2 Quality checks. . . . 122

20.1 Creating the nodestring to split elements. . . . 123
20.2 Split elements. . . . 124
20.3 Coarse Elements near the boundary. . . . 124
20.4 Feature Arc created near to the boundary to refine the mesh. . . . 125
20.5 Refine mesh. . . . 125
20.6 Nodes passing through the Feature Arc. . . . 125

21.1 Expression to calculate Froude number. 128

22.1 Export shapefile command. 129

List of Tables

5.1 Culvert data. 28

18.1 Files generated by the HEC-RAS Extraction tool. 119

1 Introduction

This document includes instructional material and tutorials to facilitate the application of the RiverFlow2D model using SMS.

RiverFlow2D is a combined hydrologic and hydrodynamic, mobile bed and pollutant transport finite-volume model for rivers, estuaries, coastal areas, and floodplains. It can simulate floods and simulate inundation over complex terrain at high resolution and with remarkable stability, accuracy and remarkable speed. The use of adaptive triangular-cell meshes enables the flow field to be resolved around key features in difficult river environments.

RiverFlow2D user interface is based upon Surface-Water Modeling Solution (SMS) developed by Aquaveo. This GIS-integrated software system provides interactive functions to generate and refine the finite-volume flexible mesh, and uses familiar GIS objects to construct a high level representation of the model. SMS facilitates assigning boundary conditions, roughness values, and other parameters, allowing the user to efficiently manage the entire modeling process. SMS also offers a comprehensive set of visualization tools including 3D rendering, animations and export graphs to Google Earth.

RiverFlow2D computation engine uses an accurate, fast and stable finite-volume solution method that eliminates the boundary and hot start difficulties of some two-dimensional flexible mesh models. RiverFlow2D also includes capability to handle hydraulic structures such as culverts, weirs, bridges, and gates. The hydrologic capabilities include spatially distributed rainfall, evaporation and infiltration. This current version also includes spatially distributed wind stress on the water surface.

2

Creating your First RiverFlow2D Project

This tutorial provides a step-by-step guide to help you setting up a new RiverFlow2D project from scratch. These guidelines include instructions for entering terrain elevation data, running the RiverFlow2D model and visualizing the model results. As an example, a river simulation with available bed elevation data is used. The boundary conditions include discharge inflow and water surface elevation outflow.

The following steps are discussed:

1. Initial steps to create a new RiverFlow2D project
   - Open SMS
   - Open RiverFlow2D template
   - Defining spatial projections and unit system
   - Read in bed elevation data
   - Adjust display options
2. Model Domain
   - Define coverage type (Generic Model)
   - Create boundary polygon
3. Define vertex spacing on arcs
4. Define polygon attributes
5. Enter boundary conditions
6. Generate mesh
7. Enter Manning's n
8. Enter Control Data
9. Run RiverFlow2D

10. Viewing RiverFlow2D Results
    - Scalar Dataset Options
    - Vector Dataset Options
11. Film Loop Visualization

All the data files required follow this tutorial are in the following directory:
`...\Documents\RiverFlow2D_SMS\ExampleProjects\Hoh`

2.1 Run SMS and Open RiverFlow2D Template

To start using RiverFlow2D, double-click the SMS icon on your desktop  .

Every time you start a new project, you must first import the RiverFlow2D template. This template has been developed by Hydronia in collaboration with Aquaveo and integrates the RiverFlow2D numerical engine into the SMS graphical user interface to pre-processing RiverFlow2D model data and post process results. To import the template file do as follows:

1. Open the SMS Program.
2. On the *File* menu, click *Open*.
3. Select the file `RiverFlow2D_Template.2dm` in the folder for this tutorial and click the *Open* button.

2.2 Define Projections and Units

In SMS you should define the coordinate projection for the site you are going to simulate. RiverFlow2D can use either Metric or English units. Also, setting a projection allows you to take advantage of all the geo-referencing tools in SMS for pre and post processing including exporting model results to Google Earth. It is a requirement to use consistent units throughout your project.

To set the Projection:

1. On the *Display* menu, click *Projection*.
2. On the *Horizontal* frame select *Global projection* and click *Set Projection...*.
3. Under *Projection*, select *State Plane Coordinate System*
4. Under *Zone* select *Washington North (FIPS 4601)*
5. Under *Datum* select *NAD83*
6. Under *Planar Units* select *Feet (U.S. Survey)*.
7. Click OK.
8. Ensure that the Vertical units are set to *U.S. Survey Feet*.

9. Click the OK.
10. On the *File* menu, click Save As... to save the project.
11. Make sure the *Save as type* is Project Files (.sms) and enter the name `HohRiver.sms`.
12. Click *Save*.

The horizontal and vertical units in RiverFlow2D need to be the same for a given project. If the horizontal projection is set to meters, the vertical units should also be in meters and so on. You could import in data with different units but you would need to convert them to the appropriate units set in your projection before running RiverFlow2D.

2.3 Read in Initial Data

The next step is to read in data to start building the model site. For the Hoh River you have a background image of the site and an elevation survey.

2.3.1 Read in Aerial Image

1. On the *File* menu, click *Open*.
2. Select the file `Hoh.tif` in the Folder for this tutorial, and click *Open*.
3. Click *Yes* if prompted to build image pyramids. This option creates images at various resolutions for clearer images as you zoom in and out of your simulation. It is especially useful with very high resolution files to improve the refresh time within SMS without losing image details.

2.3.2 Read in Topographic Data

1. On the *File* menu, click *Open*.
2. Select the file `BedElevations.txt` in the data files folder for this tutorial and click the *Open* button.
3. Select *Use Import Wizard* and Click the OK to define the *File Import Options* for the text (ASCII format) file.
4. The first step of the *File Import Wizard* gives you the option to specify delimiters and specify a starting point for importing. The defaults are fine for this data set, so click on the *Next* button.
5. Click *Finish* to import the elevations.

2.3.3 Display Options

To display the data according to the elevation values, adjust the display as follows.

1. On the *Display* menu, click *Display Options*.

2. Toggle off *Points* and select *Boundary*, then select *Contours* in the *Scatter* Display Tab.

3. Under the *Contours* tab, change the *Contour Method* to *Color Fill* and the *Transparency* to 30%.

4. Click OK to exit the *Display Options* dialog.

The resulting graph should appear in the display window similar to Figure 2.1 below.

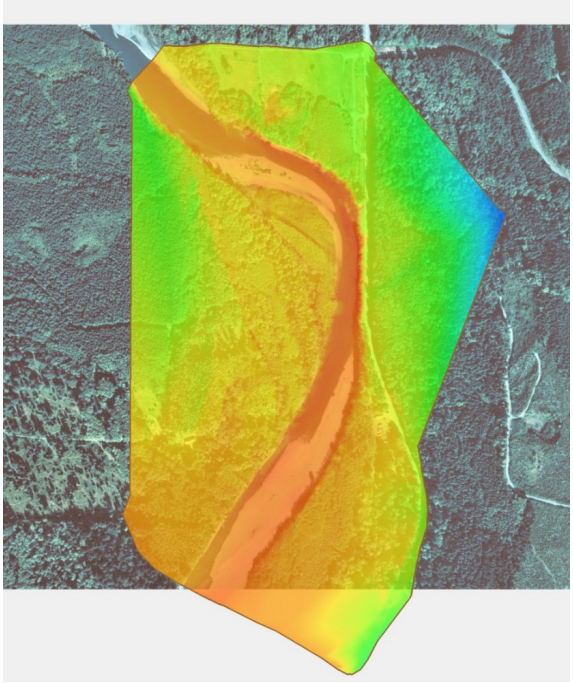


Figure 2.1 – Elevation data and background image.

2.4 Define Model Domain

2.4.1 Define Coverage Type

A coverage in the Map Module is where different model features are represented using GIS objects such as points, arcs and polygons. The shape of the modeling domain should be defined in a *Generic 2D Mesh coverage*.

1. Right click on the default coverage under *Map Data* and select *Type* | *Models* | *Generic 2D Mesh*.

2. Right click on the default coverage and select *Rename*.

3. Change the name of coverage to '*Hoh River*'.

2.4.2 Create a Polygon to Define the Model Extent

1. Select the *Create Feature Arc* Tool .

2. Begin by creating a feature arc that spans across the northern most part of the channel as shown in Figure 2.2. Click on the first vertex, move the cursor to the second vertex and double-click to end the arc.

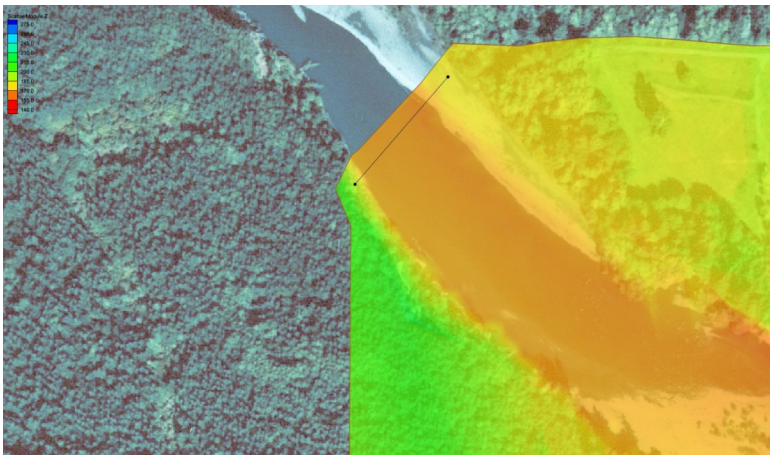


Figure 2.2 – Upstream boundary.

3. Similarly, create another arc at the southernmost part of the channel as shown in Figure 2.3.

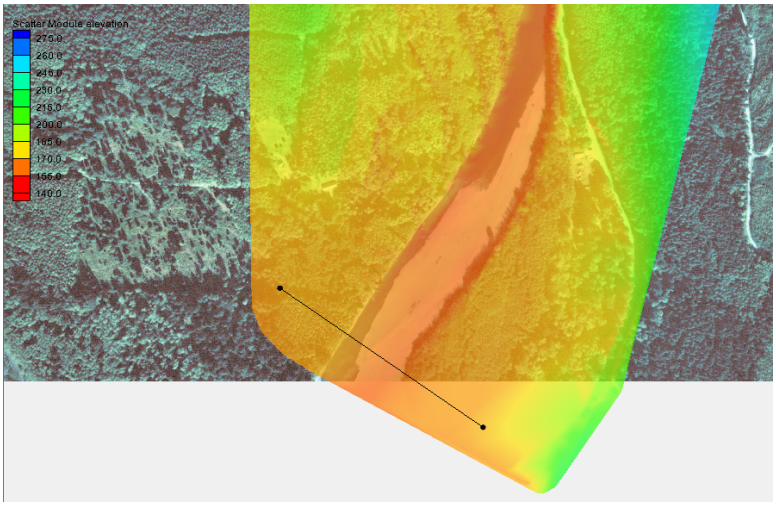


Figure 2.3 – Downstream boundary.

4. Now create feature arcs connecting the two eastern nodes and two western nodes approximately following the floodplain boundary.

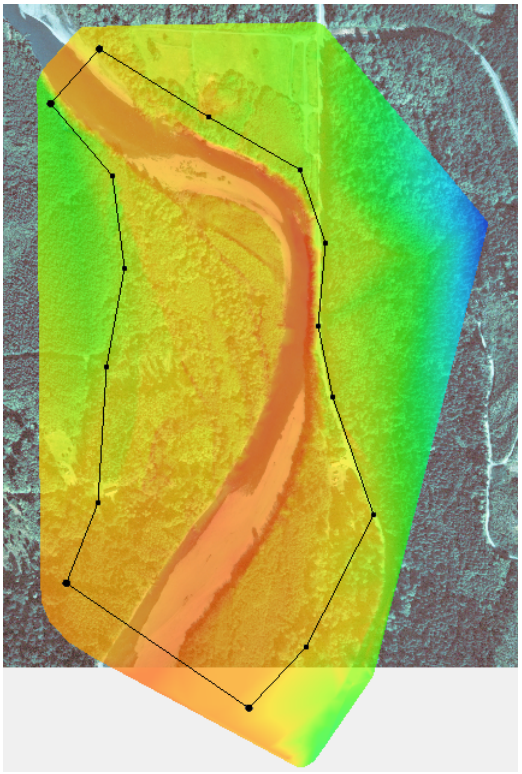


Figure 2.4 – Model extent arcs defined.

2.4.3 Define Arc Spacing

The different arcs in the Mesh Boundary coverage can be used to define the mesh cell sizes. There are different tools that you can use in SMS to control cell sizes, but in this tutorial the spacing between vertices will be used to define the cell size.

2.4.4 Redistribute Arcs Vertices

1. Select the arcs defining the East and West Banks of the reach as well as the Northern and Southern cross section arcs. To select multiple arcs, first click the *Select Feature Arc* tool and click on each arc while holding down the shift key.
2. Right-click inside the modeling area, and from the pop-up menu, click *Redistribute Vertices...*
3. In the *Arc Redistribution* section select *Specified Spacing* from the *Specify* drop down menu. Set the *Spacing to 40 ft.*
4. Click OK to exit the *Redistribute Vertices* Dialog.

The resulting domain outline with redistributed vertices is shown in Figure 2.5.

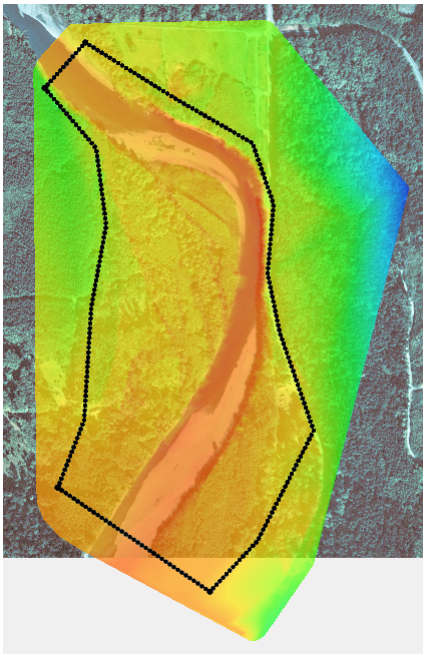


Figure 2.5 – Final Conceptual Model coverage.

2.4.5 Define Polygon Attributes

2.4.5.1 Polygon Materials (Manning's *n*)

RiverFlow2D allows you to vary the Manning's n value spatially. However, in this tutorial we will use a single Manning's n to for the whole area. In SMS you enter the n values as Polygon Materials as follows:

1. On the *Feature Object* menu, click *Build Polygons* to create polygons from the defined arcs.

   With polygons now created you can assign *Material Types* to them. First, you must specify what *Material Types* you wish to define and then assign the *Material Type* to the different polygons.

2. On the *Edit* menu, click *Material Data* to bring up the *Material Data* dialog.

3. By default there is a material named 'material 01'. Select '*material 01*' and change the name to *FloodPlain*.

4. You may define a material color and display pattern if you wish by selecting the *Pattern* button to the right of the material. Clicking on the button will allow you to change the fill pattern and selecting the dropdown arrow will allow you to change the color. For this tutorial you will leave the defaults and select OK to exit.

Further in this tutorial you will assign a Manning's n values to this material.

2.4.5.2 Other Polygon Attributes

To associate the polygons to bed elevations and mesh type, you can proceed as follows:

1. Select the *Select Feature Polygon* tool .
2. Click inside the polygon.
3. On the *Feature Object* menu, click *Attributes...* to bring up the 2D Mesh Multiple Polygon Properties Dialog.
4. On the *Mesh* drop-down list, select *Paving*.
5. On the *Bathymetry type* drop-down list, select *Scatter Set*.
6. Click the *Scatter Options...* button. In the *Scatter Set to Interpolate From* section, select the *elevation* dataset and leave all other options at the default values. These options control what dataset will be used to interpolate elevations to cell nodes.
7. Click OK to exit the *Interpolation* dialog.
8. On the *Material* drop-down list, select the *Floodplain*.
9. Click OK to exit.

2.5 Setting RiverFlow2D Boundary Conditions

Boundary conditions force the model with certain hydrodynamic conditions. For this model, a water discharge boundary condition will be specified at the upstream boundary and uniform outflow at the downstream end.

2.5.1 Feature Arc Attributes

RiverFlow2D Boundary conditions are assigned in the conceptual model by changing the feature arc attributes of the Map module. This section will describe how to assign different boundary conditions that will later be transferred automatically to the mesh.

2.5.1.1 Upstream Boundary Condition

1. Click Map Data in the *Project Explorer* to make the *Map Module* the active module.
2. Zoom in to the area shown in the Figure 2.6 near the upstream of the channel.

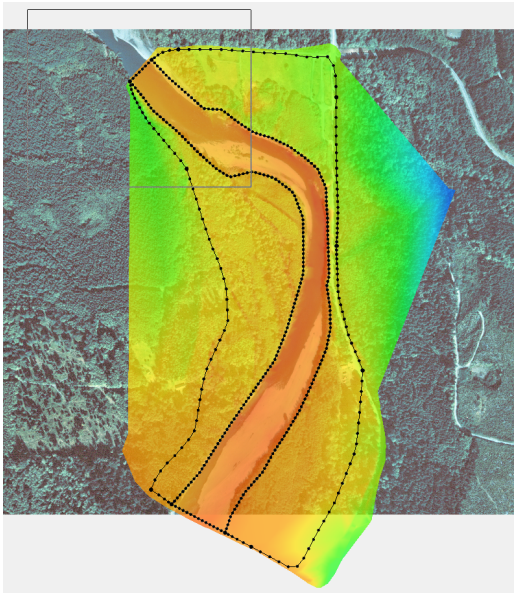


Figure 2.6 – Upstream zoomed area.

3. Using the *Select Feature Arc* Tool  select the upstream cross section arc.

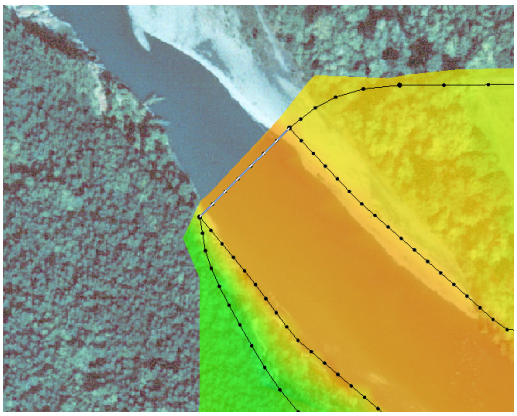


Figure 2.7 – Upstream node string.

4. With the arc selected, on the *Feature Objects* menu, click *Attributes...*
5. Select *Boundary Condition Option* and click *Options*.
6. Select *Exterior Boundary Condition* and then select *Discharge*.
7. Click *Define* and use the XY Series Editor dialog to enter the hydrograph as shown:

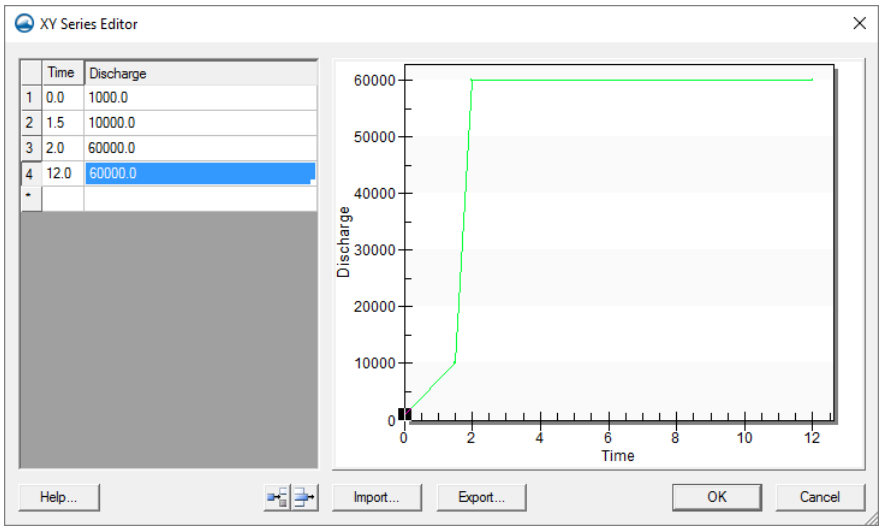


Figure 2.8 – SMS XY Series Editor used to enter hydrograph boundary condition.

8. Click OK twice to exit.

2.5.1.2 Downstream Boundary Condition

The same process applies to the downstream boundary condition.

1. Select the downstream cross section arc.
2. With the arc selected, on the *Feature Objects* menu, click *Attributes...*
3. Select *Boundary Condition Option* and click *Options*.
4. Select *Exterior Boundary Condition* and then select *Uniform Flow Outflow*.
5. Enter Energy Slope So = 0.0025
6. Click OK twice to exit.

2.6 Generate the Mesh

With the meshing parameters set, the conceptual model is ready to be converted into to a triangular-cell mesh for RiverFlow2D.

1. Select the '*Hoh River*' coverage under *Map Data* to activate it and to deselect any polygons that might be selected.
2. On the *Feature Objects* menu, click *Map* → *2D Mesh* menu item.

3. Click OK in the *2D Mesh Options* dialog leaving the default settings.

   A triangular-element mesh is created. The node elevations are interpolated values from the scatter set survey and element material types are based on the materials set in the polygons attributes. At this point, the *background image*, *scatter set* and *map data* can be turned off in the project explorer to make it easier to work with the mesh. To do this:

4. In the *Project Explorer*, uncheck the box next to the *Hoh.tif* image, the scatter set *BedElevations* and the map coverage(s).

You may want to adjust the mesh display settings to see the elevations contours.

1. Click the newly created Mesh coverage under *Mesh Data*, e.g. *Hoh River*
2. On the *Display* menu, click *Display Options*.
3. Select *2D Mesh* from the list on the left.
4. Toggle on *Contours*.
5. Under the *Contours* tab, select *Color Fill* on the *Contour Method* drop-down list.
6. Click OK to exit the *Display Options* dialog.

   The resulting image should appear similar to Figure 2.9 below.

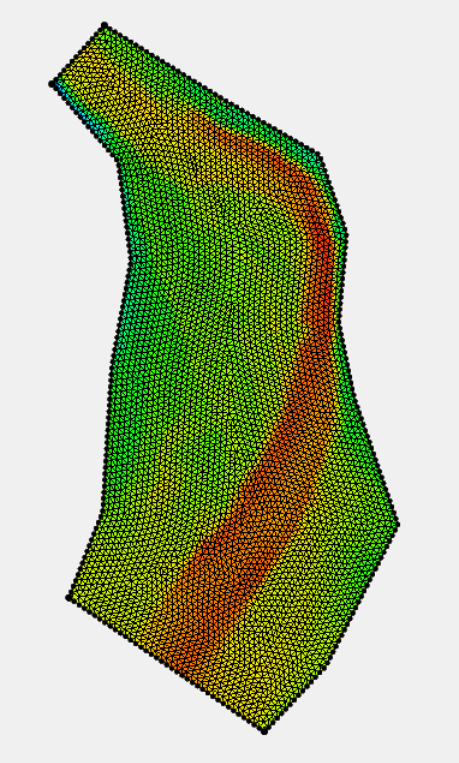


Figure 2.9 – Mesh with elevation contours.

In SMS you can also visualize the site in 3D.

7. On the *Display* menu, click *Display Options*.

8. Select *General* in the list on the left.
9. Toggle off *Auto z-mag*.
10. Change the Z magnification to 10 to improve the 3D visualization.
11. Click OK to exit the *Display Options* dialog.
12. Select the *Rotate* tool from the *Static Toolbar*.
13. Left click in the Graphics Window and drag to rotate the grid in 3D. You may also use the mouse wheel to zoom in and out. See Figure 2.10.
14. You can go back to plan view by selecting the Plan View Icon in the Display Toolbar.

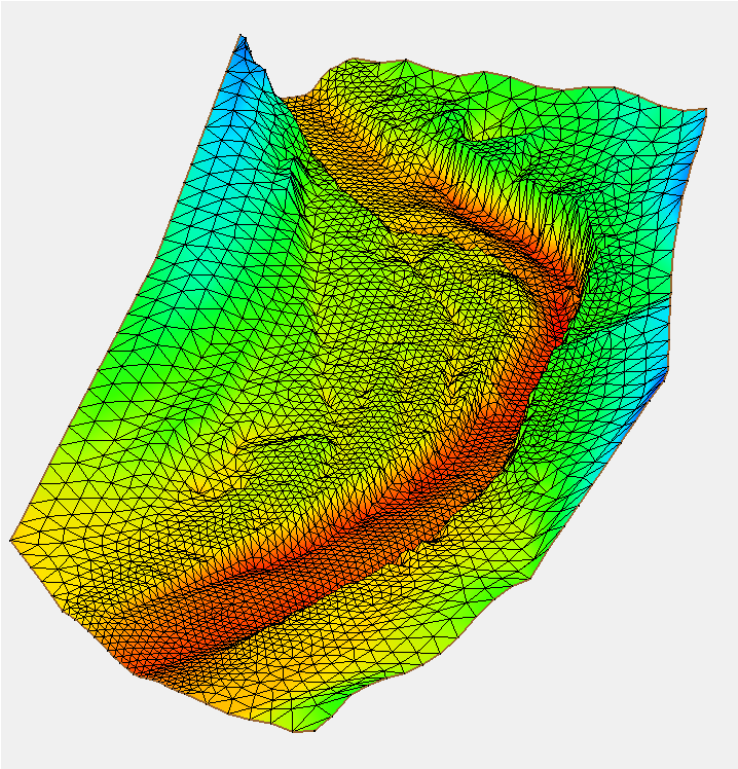


Figure 2.10 – 3D view of the mesh.

2.7 Assign Manning's *n* to Materials

To assign the corresponding Manning'n n value to the Floodplain material defined before, do as follows:

1. Click the *Mesh Data* coverage group.
2. On the ***RiverFlow2D*** menu, click *Material Properties...*
3. In the ***RiverFlow2D*** *Material Properties* dialog, select the material Floodplain.

4. Under the Control Data tab, enter a value of 0.035 for the Manning's n.
5. Click OK.
6. Save the project.

The material properties have now been properly defined.

The material zones can be displayed by opening the Display Options dialog and turning on the Materials check box.

2.8 Assign RiverFlow2D Parameters

Now that you have generated the unstructured mesh you need to set up RiverFlow2D run control data and other non-spatial model parameters.

1. Select the *Mesh Data* in the Project explorer to make the *Mesh Module* the active module.
2. On the ***RiverFlow2D*** *Menu* menu, click *Run RiverFlow2D*

   This will bring up the RiverFlow2D Data Input Program in which you can set all the run control parameters for RiverFlow2D.

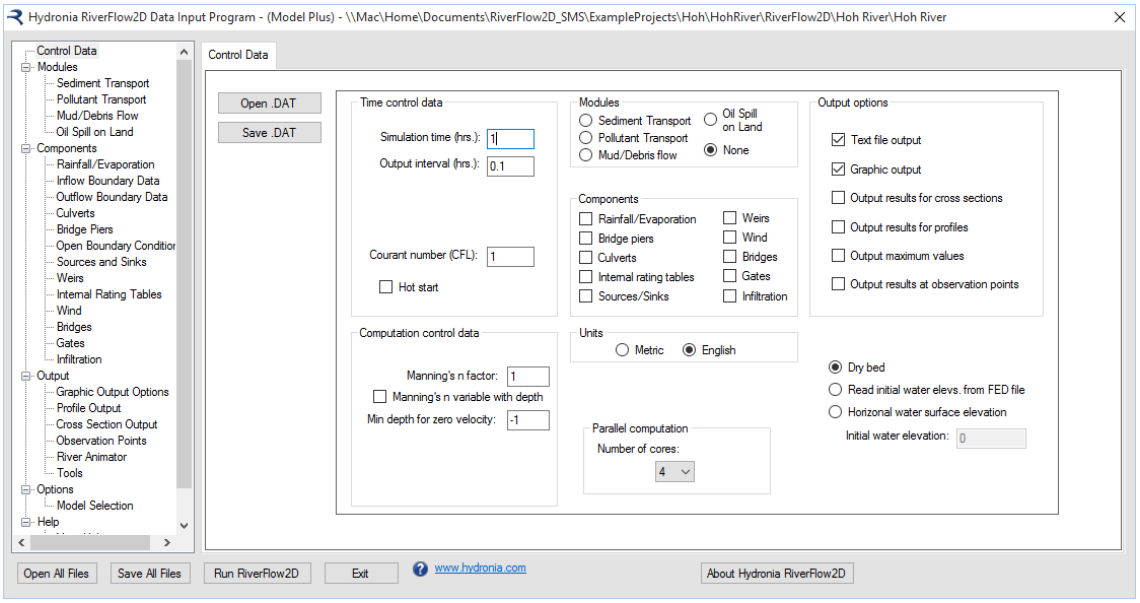


Figure 2.11 – RiverFlow2D Data Input Program main window.

3. For this tutorial accept the default values as shown in the *Control Data* panel.

2.9 Running RiverFlow2D

To run the model, click the *Run* ***RiverFlow2D*** button on the lower section of the Data Input Program.

The following window will appear indicating that the model has started running. The window also reports the simulation time, volume conservation error, total inflow and outflow discharge, and other parameters as the run progresses.

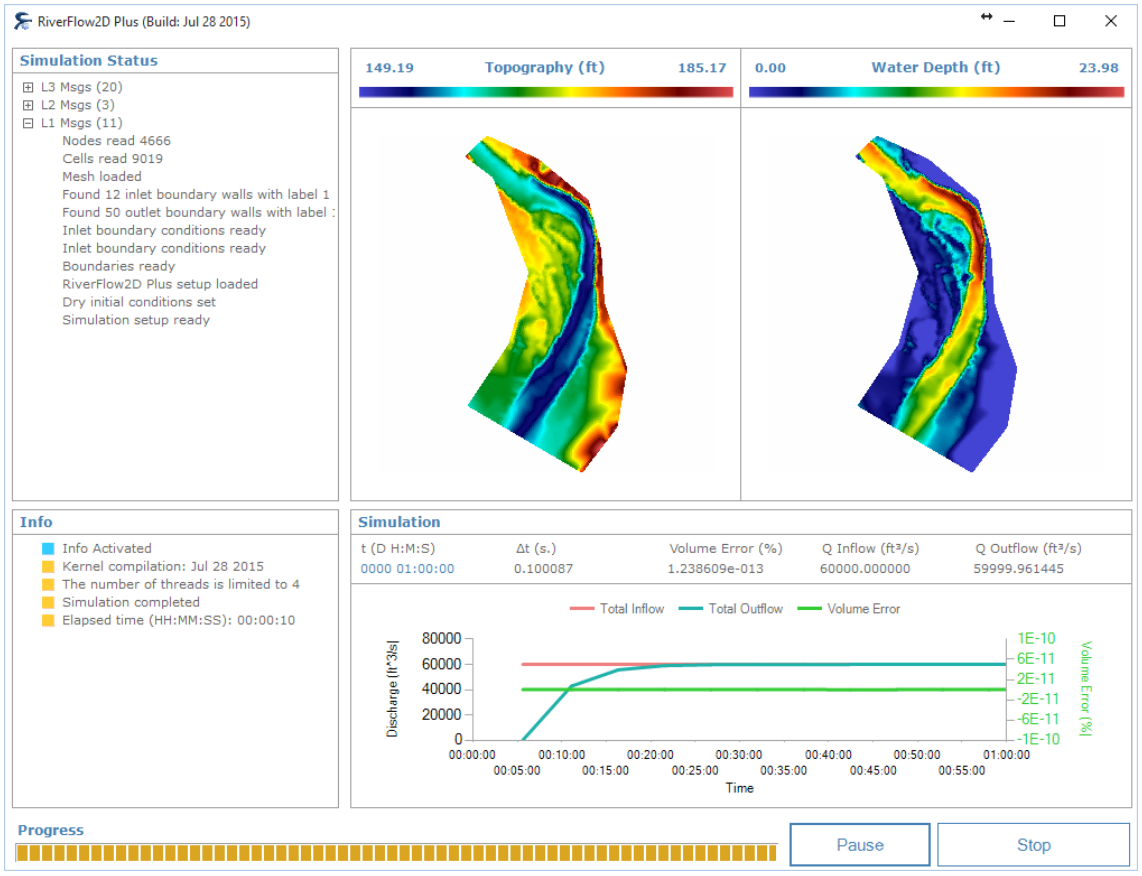


Figure 2.12 – Model run report window.

This completes the *Creating your First RiverFlow2D Project* tutorial. You may want to follow the next tutorial to learn how to visualize the RiverFlow2D results you have just generated.

3 Visualizing RiverFlow2D Results

This tutorial shows how to use several SMS tools to visualize RiverFlow2D results.

For this tutorial we will assume you have successfully completed the previous tutorial.

To visualize RiverFlow2D results, first you need to read in the XMDF output that RiverFlow2D has generated.

1. On the *File* menu, click *Open*.
2. Find the `Hoh River_results1.hdf5` file located in the folder where your project is saved and select the file. This folder will have the same name as the mesh (e.g. Hoh River) and will be three directory levels below the one containing the `.sms` file.
3. Click the *Open* button.

Once the RiverFlow2D output file has been imported as Mesh Dataset you will have many options to view the data. The Project Explorer may be used to select the desired Scalar and Vector output datasets. Scalar data sets are identified by the scalar icon and the vector datasets have the vector icon identifying them.

3.1 Scalar Dataset Options

A good way to view the output is to edit the contour display options. To change the contour properties:

1. Activate the *Water Surface Elevation* output scalar dataset by selecting it in the Project Explorer.
2. Select the *Display Options* macro in the *Display Toolbar*.

3. In the *2D Mesh tab*, click the *All off button* to turn off current display options.
4. Turn on the *Contours*, and *Mesh boundary*.
5. Under the *Contours* tab, change the *Contour Method* to *Color Fill*.
6. For the Number of contours, enter 25.
7. Click OK to exit the dialog box
8. Select Time step '0 01:00:00'.

SMS will redraw the screen similar to one shown below.

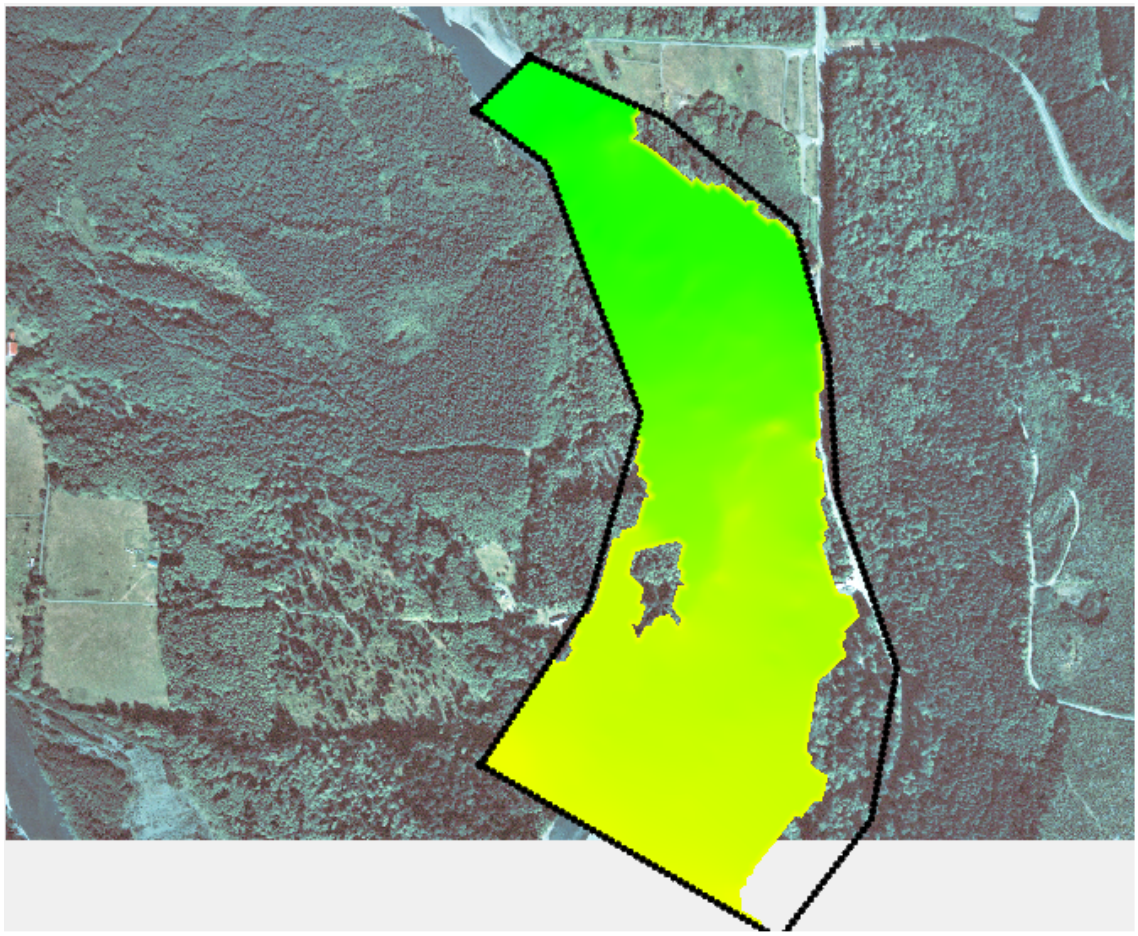


Figure 3.1 – Shaded contour plot for the Water Surface Elevation dataset.

3.1.1 Vector Dataset Options

You can display velocity vectors in this way:

1. Activate the *Velocity output vector* dataset by selecting *Velocity (ft)* in the *Project Explorer*.
2. Open the *Display Options*. In the 2D Mesh options turn on the Vectors.

3. Go to the *Vectors* tab and under *Vector Display Placement and Filter*, change the *Display* to “at each node”.

4. Enter 5 in the *Offset* and click the OK button. The *Z-offset* raises the origin of the vectors so that they are completely visible.

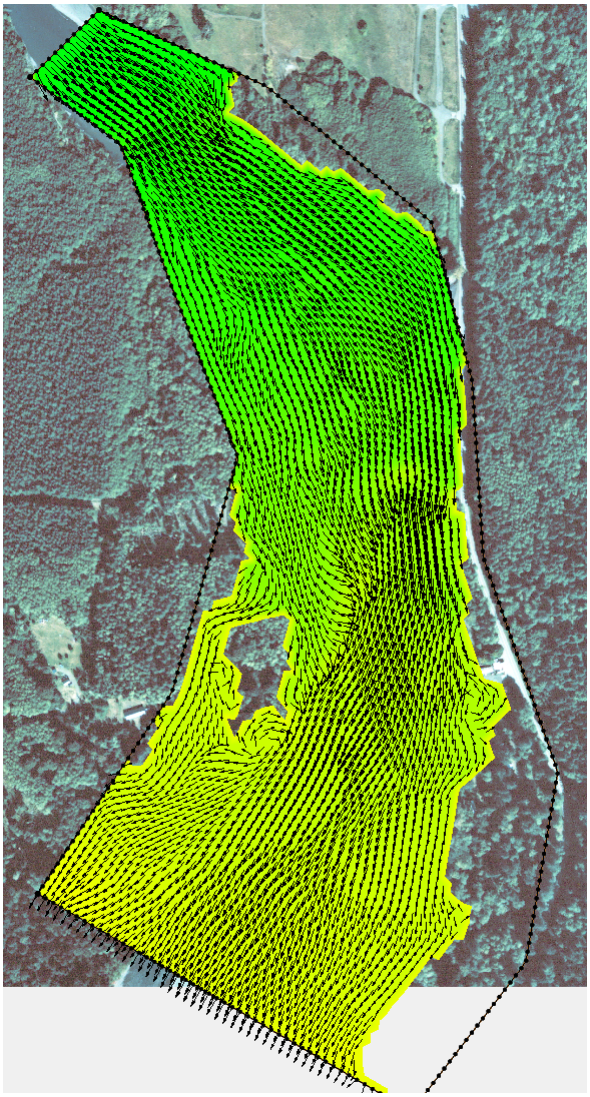


Figure 3.2 – Velocity vectors.

3.1.1.1 Film Loop Visualization

In addition to snapshots for a particular time, SMS enables the you to generate and save animations using the Film Loop. To create a film loop of the RiverFlow2D analysis:

1. On the *Data* menu, click *Film Loop...*

2. In the *Film Loop Setup* dialog, select the *Flow Trace* Type. Click the Next > button.

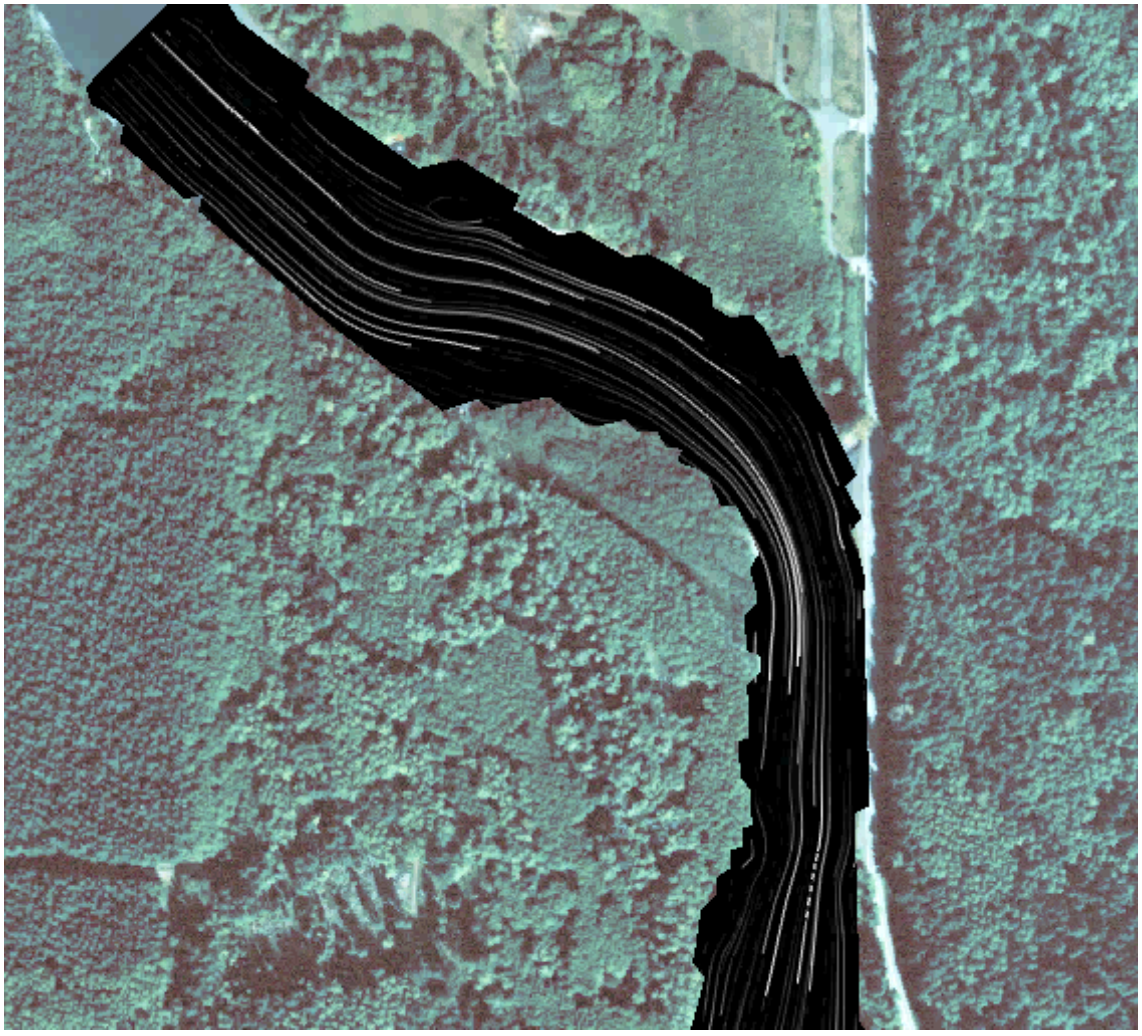


Figure 3.3 – Flow trace animation.

3. Click the Next > button twice, then the Finish button.

SMS now starts constructing the film loop adding one frame at a time. Once the last frame has been added to the loop, an AVI Application will open and the animation will start automatically. Click the Close button when finished. The film loop has been saved as `sms.avi`.

You may continue to explore other film loop features if desired. We suggest trying the *Flow Trace* option that will generate animations similar to the one depicted in Figure 3.3 and the *Create Google Earth KMZ File* to view results in the Google Earth Environment.

4 Simulating Bridges

This tutorial illustrates how to incorporate a bridge to an existing RiverFlow2D project using the *Bridges* Component. The procedure involves the following steps:

1. Create the bridge geometry data.
2. Open an existing RiverFlow2D project.
3. Enter the bridge polyline.
4. Enter bridge data.
5. Generate the mesh.
6. Run the model.

The files required to follow this tutorial can be found in the following directory:
`...\Documents\RiverFlow2D_SMS\ExampleProjects\BridgesTutorial`

4.1 Create a Bridge Geometry File

Figure 4.1 shows the front view of the bridge we want to incorporate to the model. The data is also in the `bridgegeometry.dat` file.

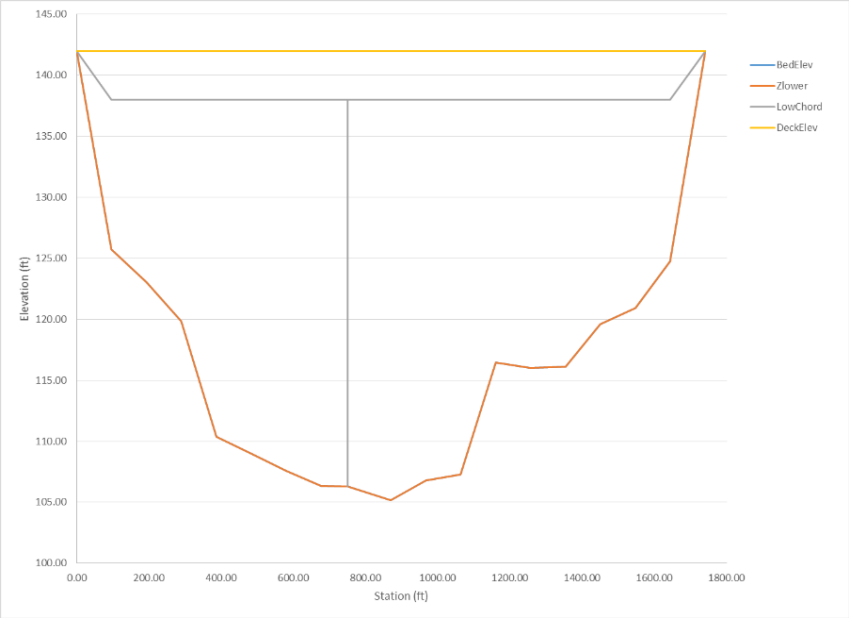


Figure 4.1 – Bridge front view.

It represents the cross section of a bridge with only one central pier, although this is just for the purpose of illustrating the Bridges component in this tutorial since the actual bridge in this location has about 12 sets of piers. This geometry is represented in the RiverFlow2D model using the bridge geometry file shown below, where the header row are presented only to describe the parameters but they should not be included in the actual data:

NP	22	Station	BedElev	Zlower	LowChord	DeckElev
		0.00	142.00	142.00	142.00	142.00
		96.68	125.72	125.72	138.00	142.00
		193.37	123.03	123.03	138.00	142.00
		290.05	119.86	119.86	138.00	142.00
		386.74	110.37	110.37	138.00	142.00
		483.42	109.00	109.00	138.00	142.00
		580.10	107.58	107.58	138.00	142.00
		676.79	106.35	106.35	138.00	142.00
		750.00	106.30	106.30	138.00	142.00
		750.00	106.30	106.30	106.30	142.00
		751.00	106.30	106.30	106.30	142.00
		751.00	106.30	106.30	138.00	142.00
		870.16	105.18	105.18	138.00	142.00
		966.84	106.77	106.77	138.00	142.00
		1063.52	107.30	107.30	138.00	142.00
		1160.21	116.47	116.47	138.00	142.00
		1256.89	116.02	116.02	138.00	142.00
		1353.58	116.09	116.09	138.00	142.00
		1450.26	119.61	119.61	138.00	142.00
		1546.94	120.92	120.92	138.00	142.00
		1643.63	124.74	124.74	138.00	142.00
		1740.31	142.00	142.00	142.00	142.00

You may use the *Bridges* panel in RiverFlow2D Data Input Program to create a bridge geometry file (see Figure 4.2). The program lets you enter data in tabular form and view a graph of the bridge geometry. You may also manipulate the graphical lines, which will make that the tabular data be modified.

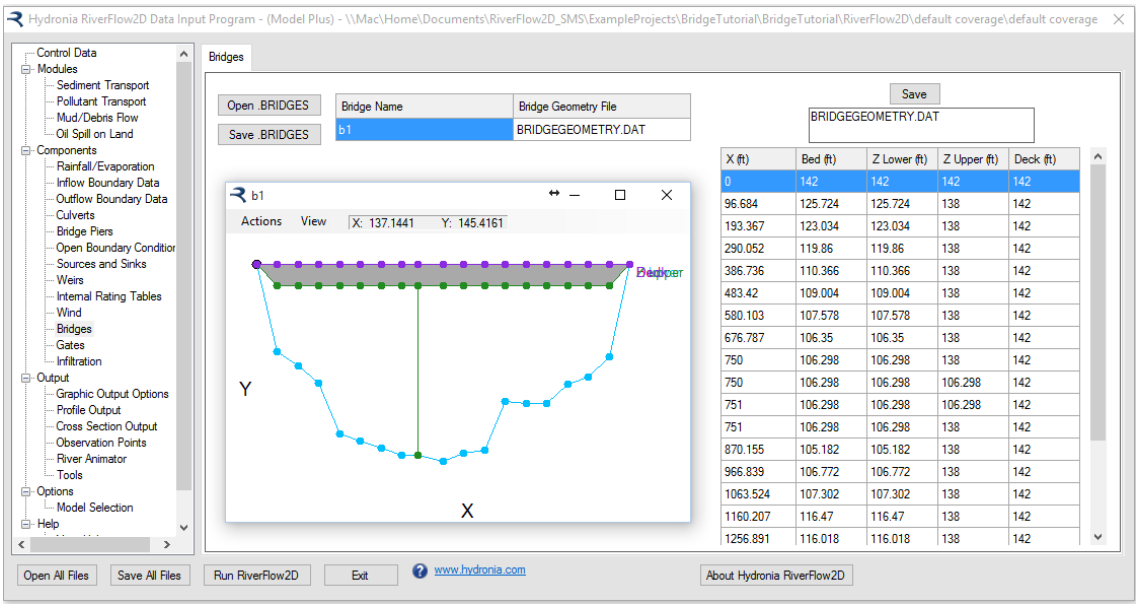


Figure 4.2 – Bridges panel in the RiverFlow2D Data Input Program.

An alternative way to create the bridge geometry file is to use a spreadsheet. In the example folder there is a MS-Excel sheet (`BridgesGeometryPlot.xlsx`) that allows editing and plotting bridge geometry files.

Before running the model you will need to manually copy the bridge geometry file to your mesh folder.

4.2 Open an Existing Project

Use the *File* | *Open* command on the SMS program to load the existing project: `BridgeTutorial.sms`. This project contains a Domain Outline, Bed Elevation data, inflow on the upper right and outflow on the lower left.

The boundary conditions are a hydrograph with a peak discharge of 220,000 cfs and *Free outflow* downstream condition.

4.3 Enter the Bridge Polyline in the Default Coverage

This step ensures that the mesh will conform to the bridge alignment, so that there will be nodes generated along the bridge. In this case we will enter the bridge is a straight line approximately 1740 ft long as follows:

1. Click on the default coverage.
2. Using the *Create Feature Arc* draw the bridge polyline on the bridge location.
3. Select the polyline using the *Select Feature Arc* and redistribute the vertices so that the spacing is 200 ft. You should see the bridge polyline as shown in Figure 4.4.

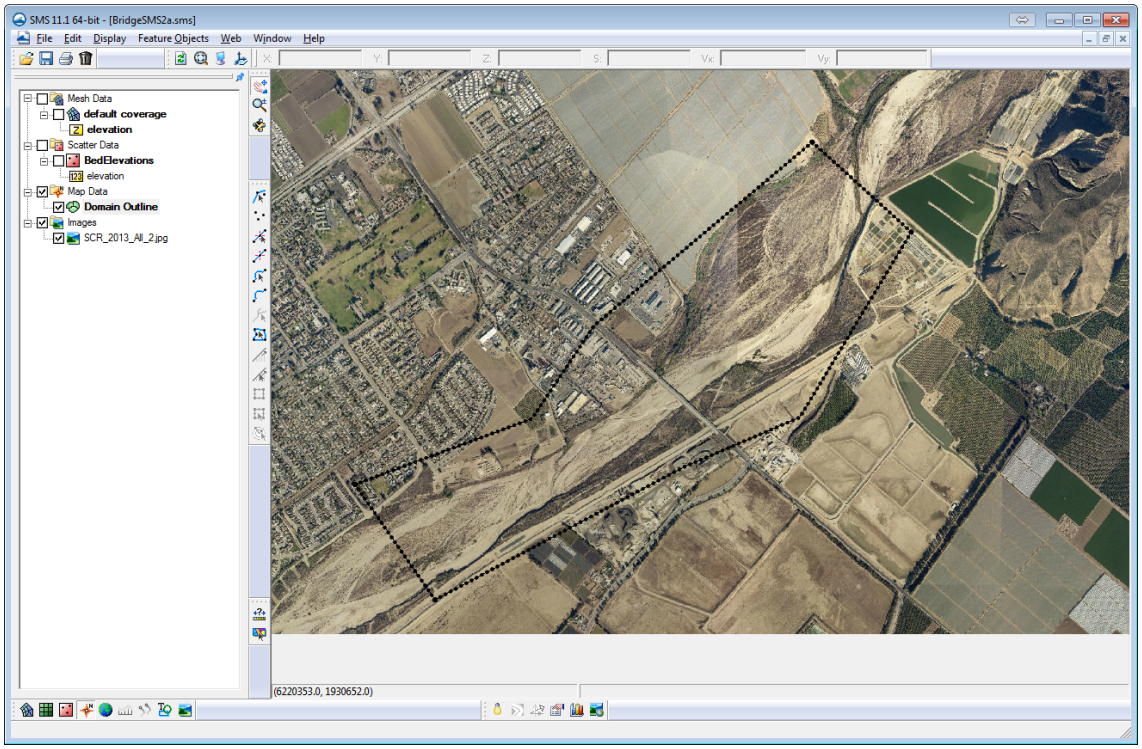


Figure 4.3 – Existing RiverFlow2D project.

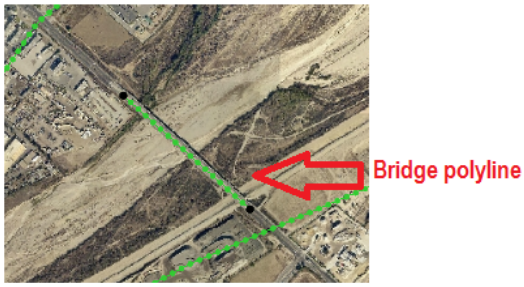


Figure 4.4 – Bridge polyline in the Domain Outline layer.

4. Now select the polyline again using the *Select Feature Arc*.

5. Right click and select *Attributes...*

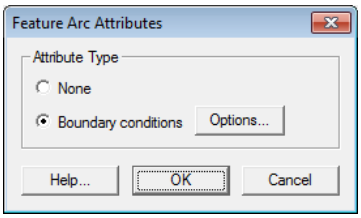


Figure 4.5 – Feature arc attributes dialog.

6. Select Boundary conditions and click Options.

7. Select *Bridge* and enter BRIDGE1 as BridgeID and `bridgegeometry.dat` for *Bridge File Name* as shown in Figure 4.6 and click OK.

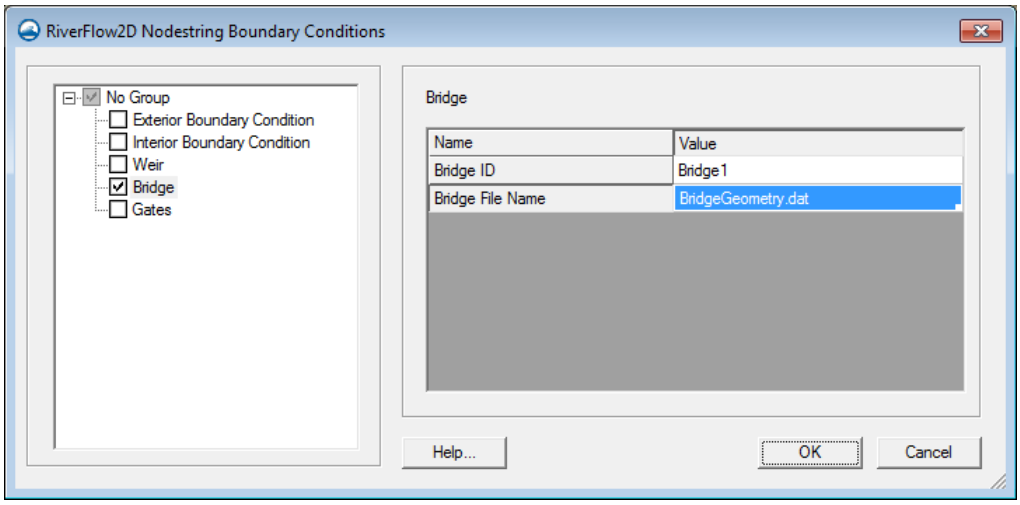


Figure 4.6 – Boundary conditions dialog.

4.4 Generate the Mesh

To finalize the spatial data preparation, use the *Map* → *2D Mesh* on the Feature Object menu to generate the mesh. Check that the resulting mesh is perfectly aligned with the bridge as shown in Figure 4.7.

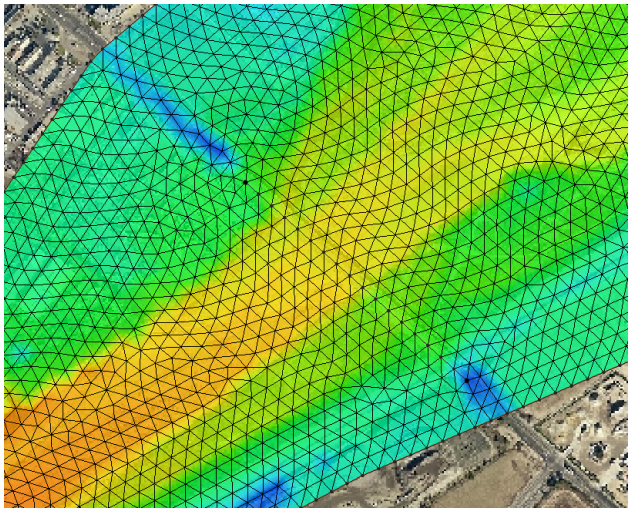


Figure 4.7 – Mesh aligned with bridge polyline.

4.5 Running the Model

Now that you have generated the unstructured mesh you need to set up RiverFlow2D run control data and other non-spatial model parameters.

Make sure to copy the `bridgegeometry.dat` file to your mesh folder before running the model.

1. Select the *Mesh Data* in the Project explorer to make the *Mesh Module* the active module.
2. On the *RiverFlow2D Menu* menu, click *Run RiverFlow2D*

   This will bring up the RiverFlow2D Data Input Program in which you can set all the run control parameters for RiverFlow2D.
3. Make sure that the *Bridges* component is selected n the *Control Data* panel:

   Leave all other parameters at their default values.

To run the model, click *Run RiverFlow2D* button on the lower section of the Data Input Program. A window will appear indicating that the model has started running. The window also reports the simulation time, volume conservation error, total inflow and outflow discharge, and other parameters as the run progresses.

5
Simulating Culverts

This tutorial demonstrates how to incorporate culverts in a RiverFlow2D two-dimensional mesh. The problem consists of a natural channel crossed by a road embankment. A culvert structure is to be used to connect the upstream and downstream parts of the channel divided by the embankment as shown in the following figure:

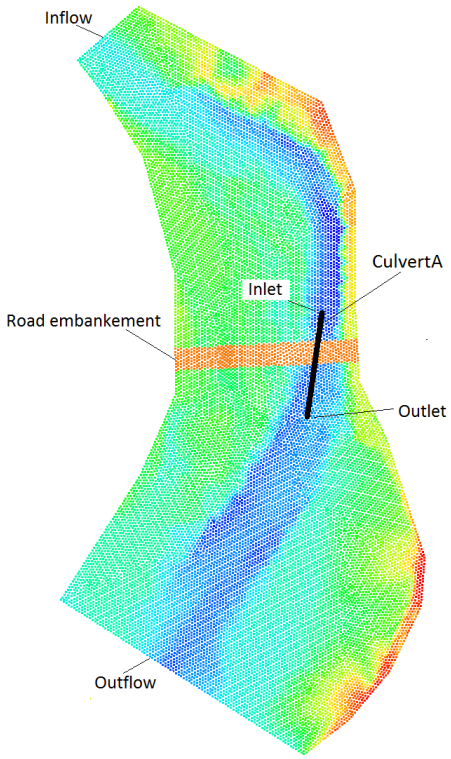


Figure 5.1 – Culvert scheme. Colors indicate the bed elevations where blue are the lowest areas and red the highest.

The water enters from upstream with a constant discharge of 1000 cfs, and outflows downstream along the indicated section. The area is initially dry.
The culvert will have a circular cross section with other characteristics summarized in Table 5.1 (CulvertA). CulvertB data is provided in case that you wanted to extend the tutorial adding a second culvert to the project.

Table 5.1 – Culvert data.

PARAMETER	DESCRIPTION	CulvertA	CulvertB
Nb	Number of identical barrels	1	1
Ke	Entrance Loss Coefficient	0.5	0.7
n	Manning's n Coefficient	0.014	0.015
Kp	Inlet Control Coefficient	0.3	0.4
M	Inlet Control Coefficient	2	2
Cp	Inlet Control Coefficient	1.28	1.1
Y	Inlet Control Coefficient	0.67	0.69
m	Inlet form coefficient	-0.5	-0.5
Dc	Diameter (ft)	3.0	2.0
–	Inlet Invert Elevation	-9999.0	-9999.0
–	Outlet Invert Elevation	-9999.0	-9999.0

The procedure to integrate this culvert in a RiverFlow2D simulation involves the following steps:

1. Open an existing RiverFlow2D Project.
2. Create new culvert inlet and outlet nodes in the Map Coverage.
3. Generate the mesh to ensure that the culvert are properly indentified.
4. Export the data and setup model parameters.
5. Run the model.
6. Review culvert output files.

The files required to follow this tutorial can be found in the following directory:
`...\Documents\RiverFlow2D_SMS\ExampleProjects\Culverts`

5.1 Open an Existing RiverFlow2D Project

Run SMS and using the *Open* command on the *File* menu, open the `CulvertTurorial.sms` file as shown in the following figure:

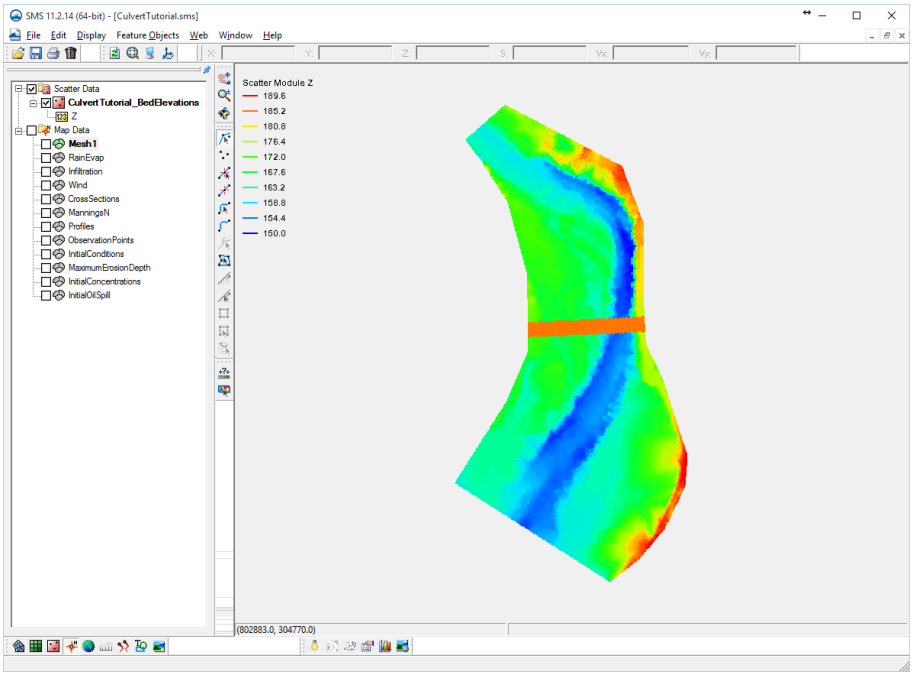


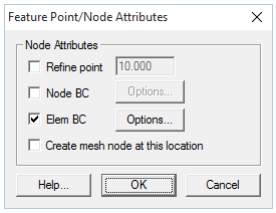
Figure 5.2 – Culvert tutorial. Initial window.

The embankment has been included in the bed elevation scatter set and is visible in the middle of the area. The project also includes an upstream discharge boundary condition and uniform flow downstream.

5.1.1 Create the Culvert

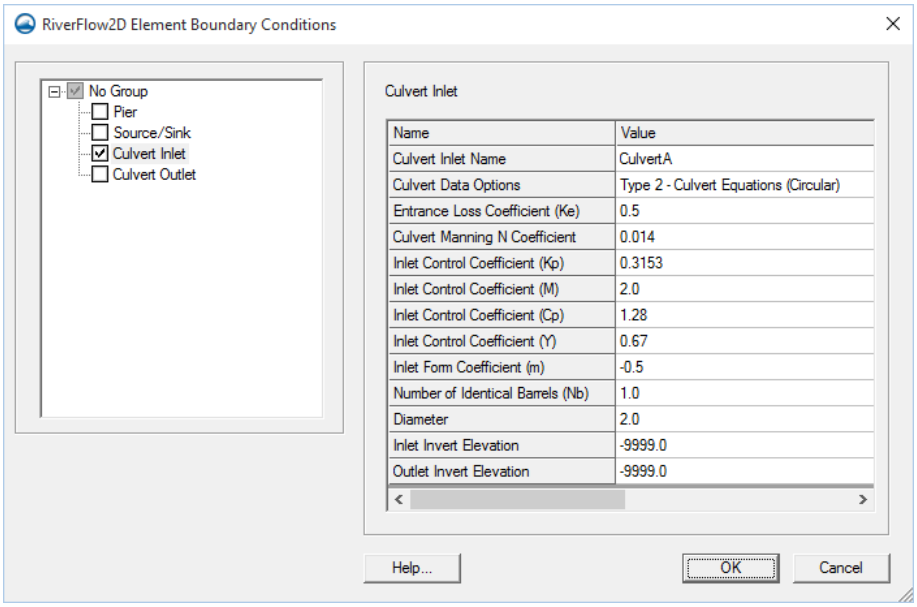
The culvert data is entered in the SMS conceptual model coverage:

1. Click on the *Create Feature Point* icon and click on the location where you want to enter the inlet point.
2. Click on the location where you want to enter the outlet point
3. Now click on the Select *Feature Point* icon
4. Click and then right-click on the point entered in the previous step.
5. On the *Feature Objects* menu, click *Node Attributes...*
6. Select *Elem BC* and click *Options...* button. Unselect *Create mesh node at the location*:



7. Click *Options...*

8. Select *Culvert Inlet*, enter the Culvert ID name (CulvertA) and enter the culvert data as shown

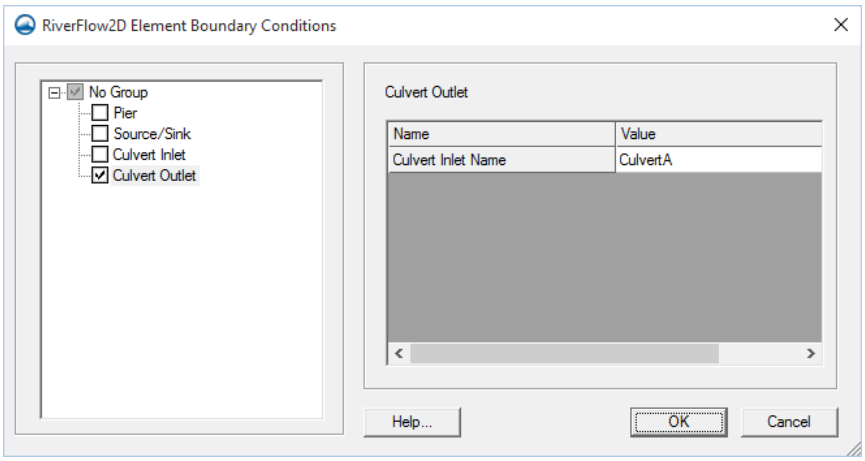


9. Click *OK* two times to close the dialogs

10. Go back to the location where you entered the Culvert Outlet Element, click and then right-click on it, and on the *Feature Objects* menu, click *Node Attributes...*

11. Select *Elem BC* and click *Options...* button. Unselect *Create mesh node at the location* and click *Options...* as you did with the inlet node.

12. Select *Culvert Inlet*, and enter the Culvert Inlet Name exactly the same as the inlet name: CulvertA



Note that which point you select as inlet and which as outlet is irrelevant. The model will dynamically make this assignment during computation, based on the actual flow direction at any given time.

Repeat the culvert creation process as above for a second culvert (CulvertB) if desired.

To save the data to the project (.2dm file), on the *File* menu, click *Save Project*.

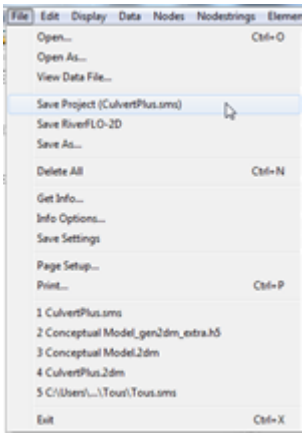


Figure 5.3 – Save data to .2dm file.

It is always recommended to re-generate the mesh after inputting the culverts. This will ensure that SMS will be correctly integrate the culvert data to the mesh.

5.2 Setup Control Parameters

With the Mesh layer active, select RiverFlow2D menu and click on *Global Parameters*. Then select the *Control Data* tab:

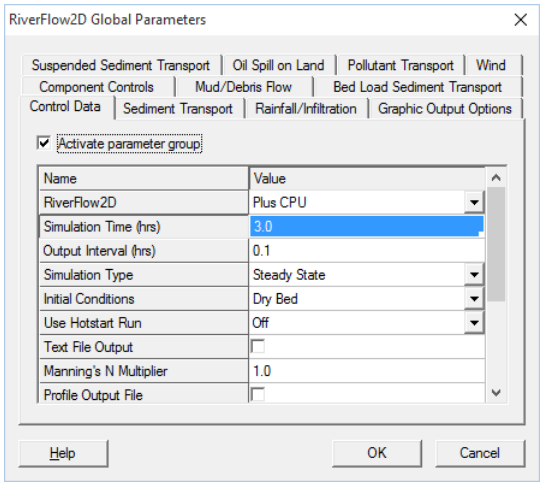


Figure 5.4 – Control data.

Now select the *Components Controls* tab and then the *Culverts component* :

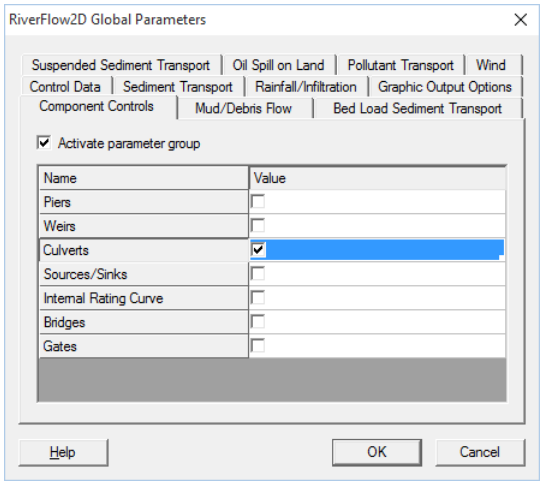


Figure 5.5 – Control data.

5.3 Running the Model

To run the model, click on the *Run* ***RiverFlow2D***. On the Data Input program, click the again *Run* ***RiverFlow2D*** and follow screen instructions. During model run, you will see the graphic output screen reporting on different parameters of the run:

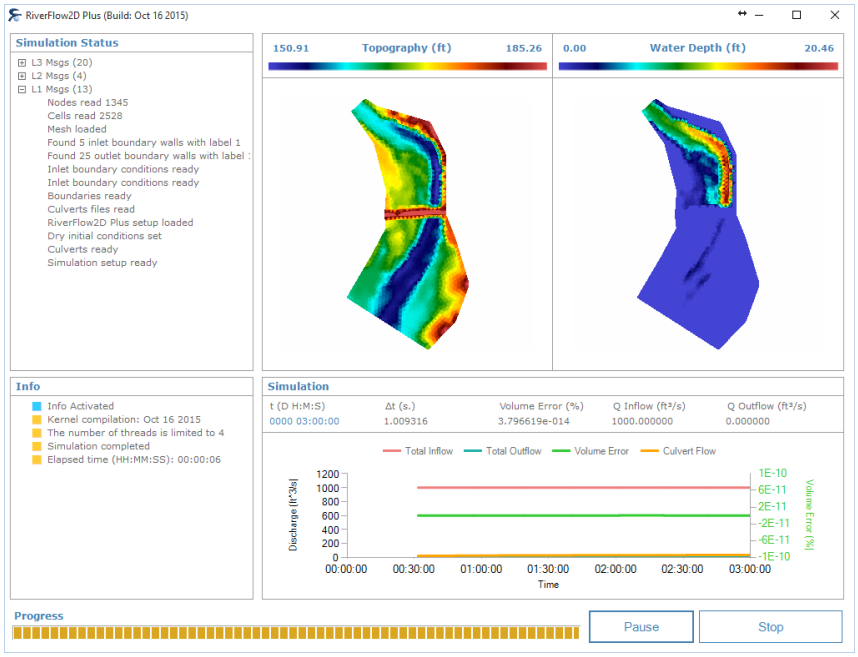


Figure 5.6 – Graphic output.

If after the first run you want to change any culvert parameter, it is better to do so in the Data Input Program and not in the SMS dialogs. The data in SMS dialogs will be erased every time you regenerate the mesh, while you will be always be able to keep the non-spatial data (e.g. culvert parameters) using the Data Input Program.

5.4 Review Culvert Output Files

For each culvert, RiverFlow2D creates an output file named: `CULVERT_culvertID.OUT`, where culvertID is the text ID entered when we created to the culverts. Each file contains the time series of discharge through the culvert, and water surface elevations at the inlet and outlet locations. For this tutorial you will fine one file named `CULVERT_CulvertA.OUT` a shown in the following figure.

```
================================================================================
            TWO-DIMENSIONAL FINITE VOLUME RIVER DYNAMICS MODEL
                      (C) COPYRIGHT 2009-2015 Hydronia, LLC.
                                    ALL RIGHTS RESERVED
                                    RUN DATE: 28/Oct/2015
================================================================================

================================================================================
Results for Culvert no.:                1 Culvert ID: CulvertA
================================================================================
 Time       Qc       WSEL1     WSEL2
 hrs.     ft3/s      ft.       ft.
0.01017   0.000   152.662    155.583
0.18018   0.000   152.682    155.583
0.19005   0.000   154.283    155.583
0.20000   0.000   155.246    155.583
0.21006   0.000   155.480    155.583
0.22011   4.593   155.900    155.612
0.23008   5.870   156.113    155.647
0.24009   7.920   156.504    155.664
0.25020   9.158   156.777    155.673
0.26027   9.862   156.956    155.678
0.27036   10.714   157.193    155.683
0.28032   11.371   157.380    155.686
0.29018   11.906   157.548    155.689
0.30000   12.543   157.745    155.693
0.31015   12.969   157.895    155.696
0.32017   13.391   158.044    155.698
0.33007   13.790   158.186    155.700
```

Figure 5.7 – Output file for Culvert A.

6 Simulating Levees using Weirs

This tutorial illustrates how to incorporate levee to an existing RiverFlow2D project using the Weirs Component. The are several advantages in using this component to simulate levees, walls, and sound walls:

- You do not need to refine the mesh to account for the high elevations along the levee/wall.
- Can account for vertical walls.
- More accurate computations of the discharge over the structure in case of flow overtopping.

The procedure to use this component involves the following steps:

1. Open a RiverFlow2D project.
2. Import a weir breakline containing elevations.
3. Convert weir breakline to the map coverage.
4. Interpolate the weir elevations.
5. Generate the mesh.
6. Run the model.

The files required to follow this tutorial can be found in the following directory:
`...\Documents\RiverFlow2D_SMS\ExampleProjects\WeirsTutorial`

6.1 Open an Existing Project

On the *File* menu, click *Open* to load `WeirsTutorial.sms`. This project contains a polygon that defines the model domain outline, bed elevation data, inflow discharge boundary condition upstream on the upper right side and outflow on the lower left.



Figure 6.1 – Existing RiverFlow2D project.

The inflow boundary conditions is a hydrograph reaching 200,000 cfs upstream and the outflow condition is set to *Uniform Flow Outflow* downstream.

6.2 Import a Weir Breakline

We want to use surveyed elevations along a polyline to assign them as weir crest elevations of a hypothetical levee that will be constructed on the right bank of the river.

You can import breaklines in SMS along with scatter data using the *File Import Wizard*. In order to import breaklines, the tabular file must be prepared in one of two supported formats. The one used in this tutorial is defined by its vertices (X, Y, Z) where Z represent the crest elevations. The breakline file is `LeveeRB_Breakline.txt` and its content is as follows:

xcoord	ycoord	zcoord	name
6219162.97	1928648.44	155.51	leveeRB
6218337.23	1927885.62	151.14	leveeRB
6217873.24	1927256.48	150.06	leveeRB
6217495.76	1926454.34	150.69	leveeRB
6216764.39	1925471.31	156.16	leveeRB
6215962.25	1924787.13	155.34	leveeRB
6215065.73	1924299.55	154.52	leveeRB
6213972.61	1923749.06	155.84	leveeRB

An additional column of data defines the breakline name that is leveeRB in this example. In the import wizard, this column should be mapped as *Breakline* as explained below.

1. On *File* menu, click *Open* and select `LeveeRB_Breakline.txt`.
2. In the *Open File Format* dialog select *Use Import Wizard* and click OK.

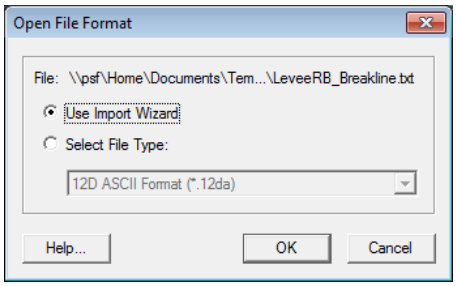


Figure 6.2 – Open File Format dialog.

3. In the *File Import Wizard* dialog select *Space* as shown and click *Next*.

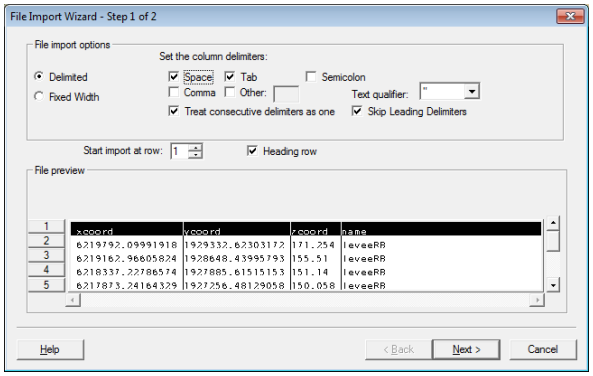


Figure 6.3 – File Import Wizard dialog step 1 of 2.

4. Select *Breakline* in the 4th column drop down list as shown in Figure 6.4 and Select *Names* as Breakline Delimiters as in Figure 6.5.

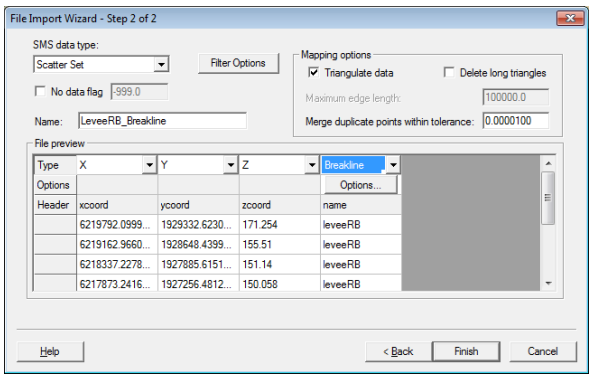


Figure 6.4 – File Import Wizard dialog step 2 of 2.

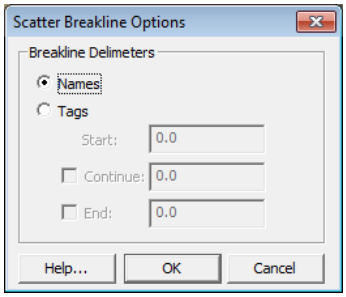


Figure 6.5 – Scatter Breakline Options dialog.

The breakline will be imported and a new Levee_RB_Breakline scatter dataset will be created.

6.3 Convert the Weir Breakline to the Map Coverage

This step is necessary to ensure that the mesh will conform to the weir alignment, in such a way that there will be nodes along the breakline.

1. To transfer the imported breakline to the Map Coverage, right-click on the LeveeRB_Breakline scatter,
2. Then click on *Convert...* and click on *Scatter Breaklines* → *Map* as shown in Figure 6.6.

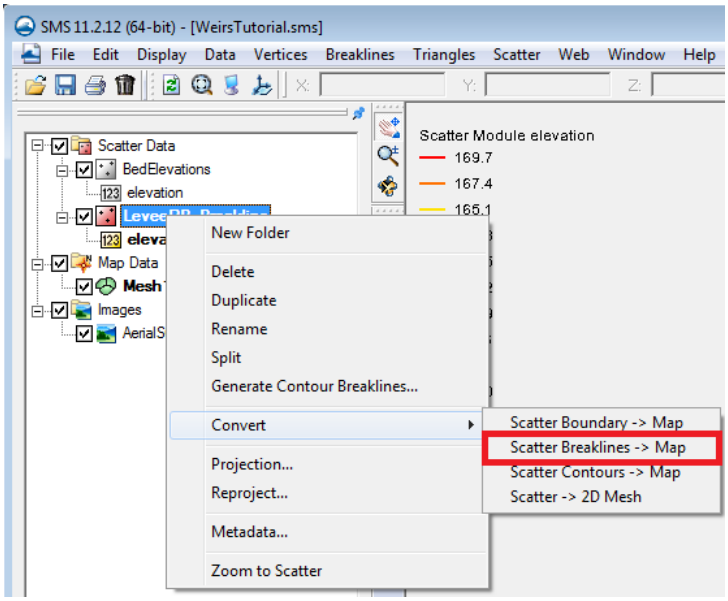


Figure 6.6 – Scatter Breaklines to Map conversion command.

3. To complete transferring the weir crest elevations, in the *Scatter* Menu click *Interpolate to Map*:

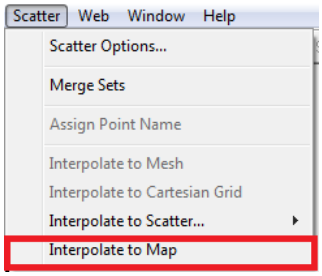


Figure 6.7 – Interpolate to Map Command.

4. Now select the Mesh1 coverage under Map Data, and using the *Select Feature Arc*, first click and then then right-click on the weir breakline.

5. Click on *Redistribute vertices...* on the pop-up menu, and in the dialog enter a node spacing of 200 ft. You should see the weir polyline as shown in Figure 6.8.

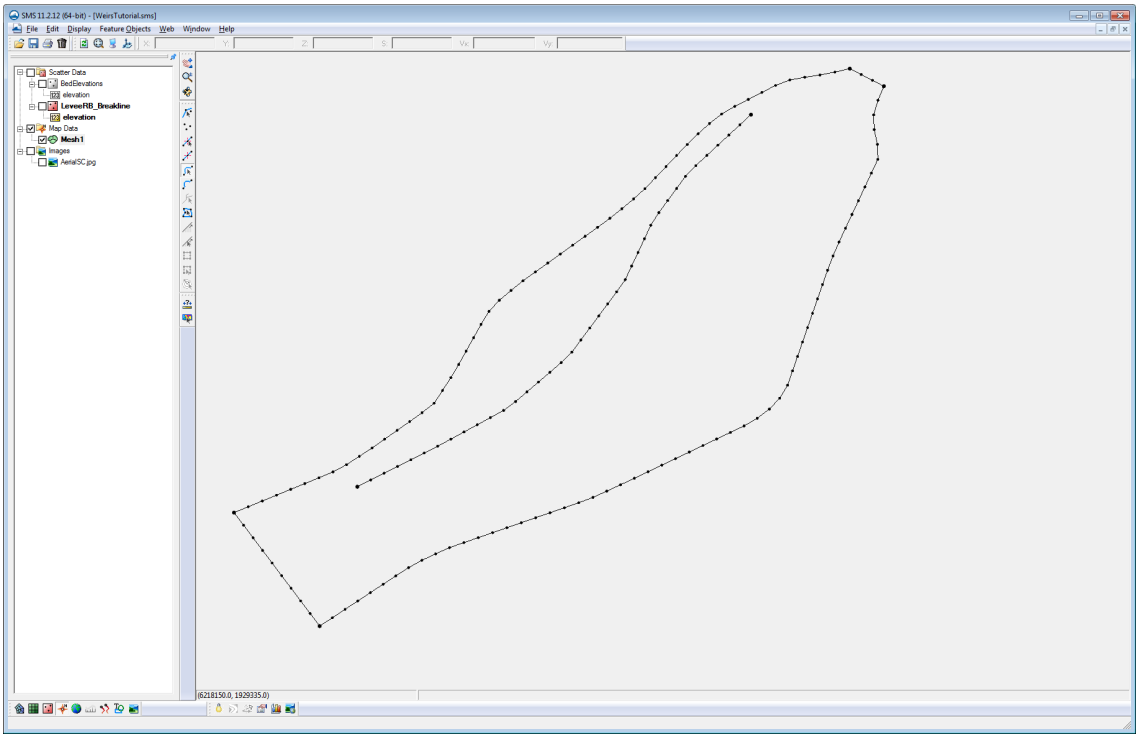


Figure 6.8 – Map data showing domain boundary and internal breakline.

6. Now right-click again on the weir breakline and in the pop-up menu, click *Attributes*. Then select *Boundary conditions* and click *Options...*

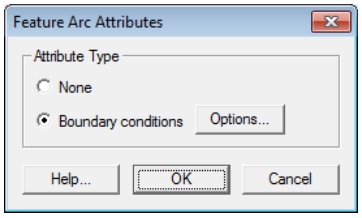


Figure 6.9 – Feature Arc Attributes dialog.

7. In the *Nodestring Boundary Conditions* dialog select *Weir* and enter the weir name: LeveeRB and *Weir Coefficient* 3.2 as shown in Figure 6.10. The *Crest Elevation* parameter is not used since the elevations will be taken from the imported breakline crest elevations.

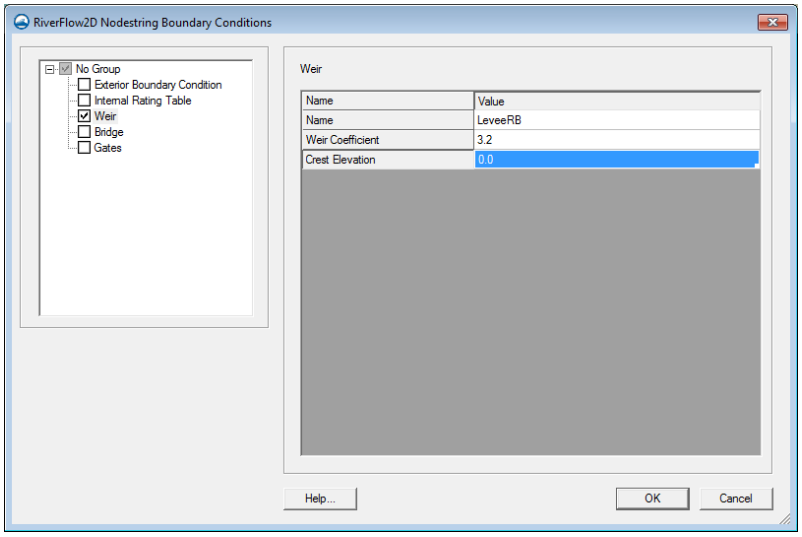


Figure 6.10 – Weir Boundary Conditions data.

6.4 Generate the Mesh

To finalize the spatial data preparation, click *Map* → *2D Mesh* on the *Feature Object* menu to generate the mesh. Check that the resulting mesh is perfectly aligned with the weir:

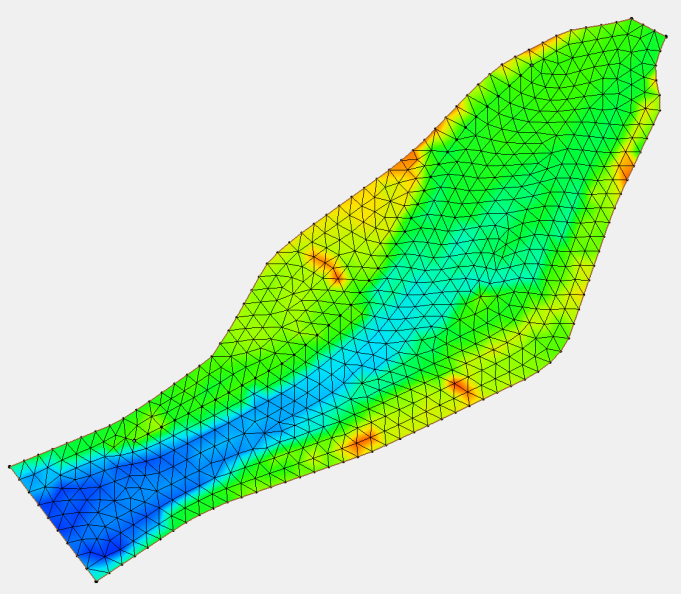


Figure 6.11 – Mesh aligned with weir polyline.

6.5 Running the Model

To run the model do as follows:

1. Select the *Mesh Data* in the Project explorer to make the *Mesh Module* the active module.
2. On the ***RiverFlow2D*** *Menu* menu, click *Run RiverFlow2D*

   This will bring up the RiverFlow2D Data Input Program in which you can set all the run control parameters for RiverFlow2D.
3. Select the *Weirs* component

   Leave all other parameters at their default
4. Finally, click the *Run* ***RiverFlow2D*** button on the lower section of the Data Input Program.

   When the model finishes the execution, you can use the visualization tools to explore results.

7 Hydrologic Simulations

The conceptual model of a hydrologic simulation with RiverFlow2D requires a series of *non-overlapping* polygons that organize the input rainfall/evaporation and infiltration data to the model. Only areas covered by polygons will receive rainfall or consider infiltration depending on the case. Each Rainfall/Evaporation polygon should be associated with a file containing a rainfall and evaporation time-series file. Similarly, each Infiltration polygon should correspond with a file containing the infiltration calculation method and its parameters for the polygon. The user will need to generate these rainfall and infiltration data files associated with each polygon, and copy them to the project folder, prior to running the model.

This tutorial illustrates how to perform a hydrologic simulation accounting for rainfall, evaporation and infiltration. The procedure includes the following steps:

1. Create new Materials corresponding to rainfall and infiltration files.
2. Create the rainfall and evaporation time-series data files.
3. Create the infiltration data files.
4. Open an existing RiverFlow2D project.
5. Create *RainEvap* coverage and the rainfall/evaporation polygons.
6. Create the *Infiltration* coverage and the infiltration polygons.
7. Generate the mesh.
8. Run the model.
9. Visualize model results.

The files required to follow this tutorial can be found in the following directory:
`...\Documents\RiverFlow2D_SMS\ExampleProjects\RainfallInfiltrationTutorial`

7.1 Open an Existing Project

On the *File* menu, click *Open* to open the existing project: `RI_TUT2.sms`.

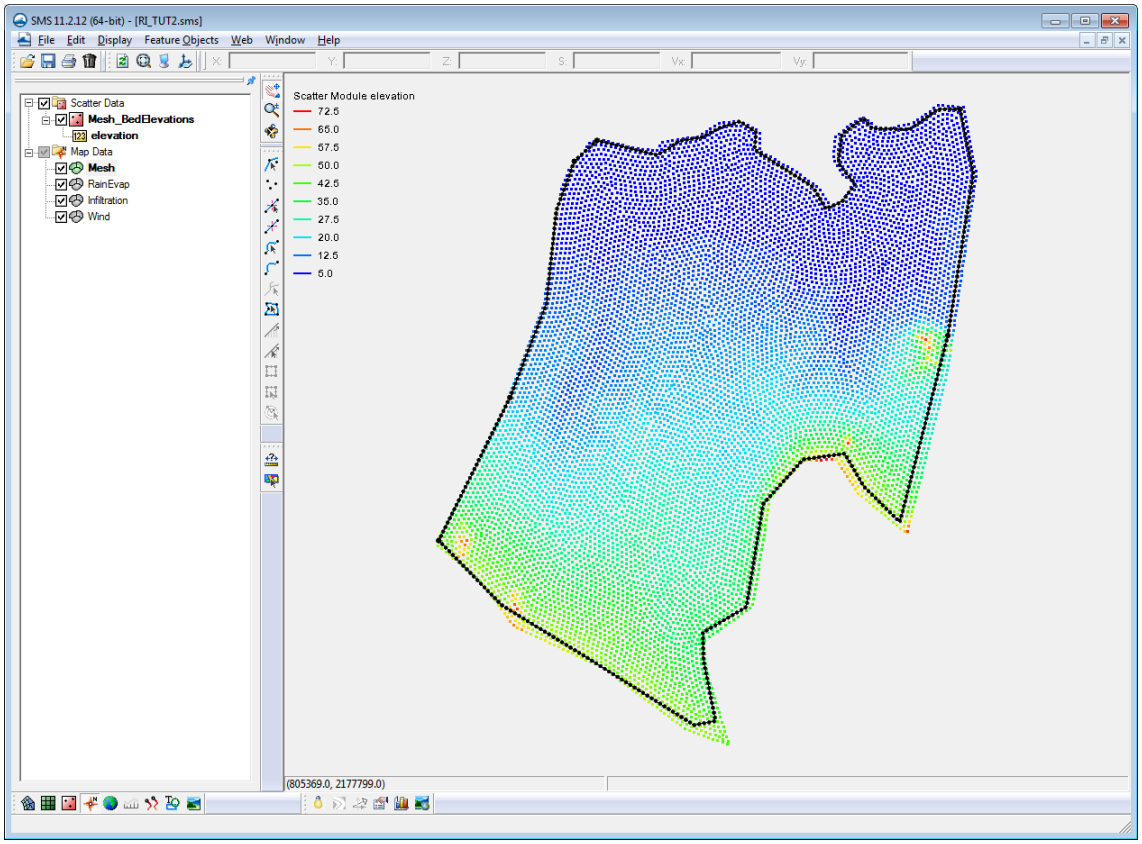


Figure 7.1 – Existing RiverFlow2D project.

This project already contains a terrain elevation dataset, a model domain extent polygon, outflow boundary conditions, and empty rainfall/evaporation (RainEvap) and infiltration (Infiltration) area coverages.

7.2 Create New Materials Corresponding to Rainfall and Infiltration Files

1. Use the *Material Data* command in the *Edit* menu to create new *Material Data* entries. In the RiverFlow2D SMS GUI the Rainfall/Evaporation and Infiltration data file names will correspond to the material names entered in the *Materials Data Dialog*.

2. To add new data file names, use the *New* button, and enter Material names that will correspond with the rainfall/evaporation and infiltration data files that you want to associate with each polygon. In this tutorial we will enter `Rainfall1.dat`, `Infiltration1.dat` and `Infiltration2.dat` as shown:

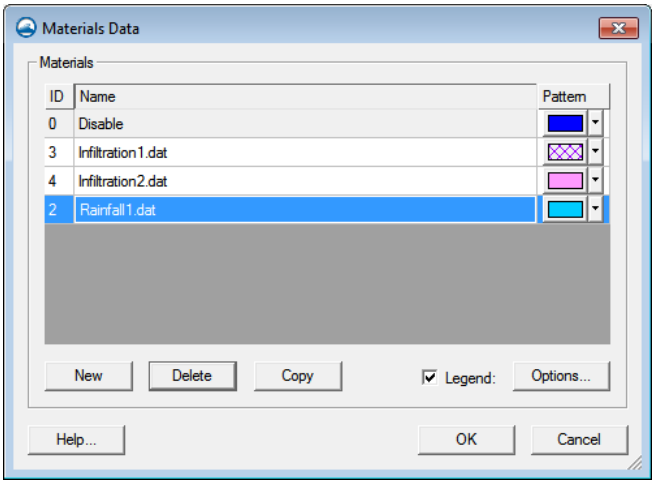


Figure 7.2 – Materials Data Dialog to enter Rainfall/Evaporation and Infiltration data file names.

7.3 Create the Rainfall and Evaporation Time-series Data File

To run a hydrologic simulation with RiverFlow2D you will create polygons over which the rainfall/evaporation data will be applied, but first you need to create the rainfall and evaporation time series ASCII text files that will be associated with each polygon. These files can be created with any text editor such as Notepad or Wordpad. The rainfall/evaporation file has the following format:

Line 1: Number of points in the time series of rainfall and evaporation.

NPRE

NPRE lines containing:

TIME Rainfall (mm/h or in/h) Evaporation (mm/h or in/h)

The following table is an excerpt of the `Rainfall1.dat` file that is included in the folder for this tutorial. In this example, the evaporation is assumed to be 0 for all times.
Figure 7.3 depicts the graphical representation of the rainfall time series in `Rainfall1.dat` file.

49		
0.00	0.00	0.00
0.50	1.53	0.00
1.00	1.53	0.00
1.50	1.53	0.00
2.00	1.53	0.00
2.50	1.84	0.00
3.00	1.84	0.00
...		
21.50	1.53	0.00
22.00	1.84	0.00
22.50	1.53	0.00
23.00	1.53	0.00
23.50	1.84	0.00
24.00	1.53	0.00

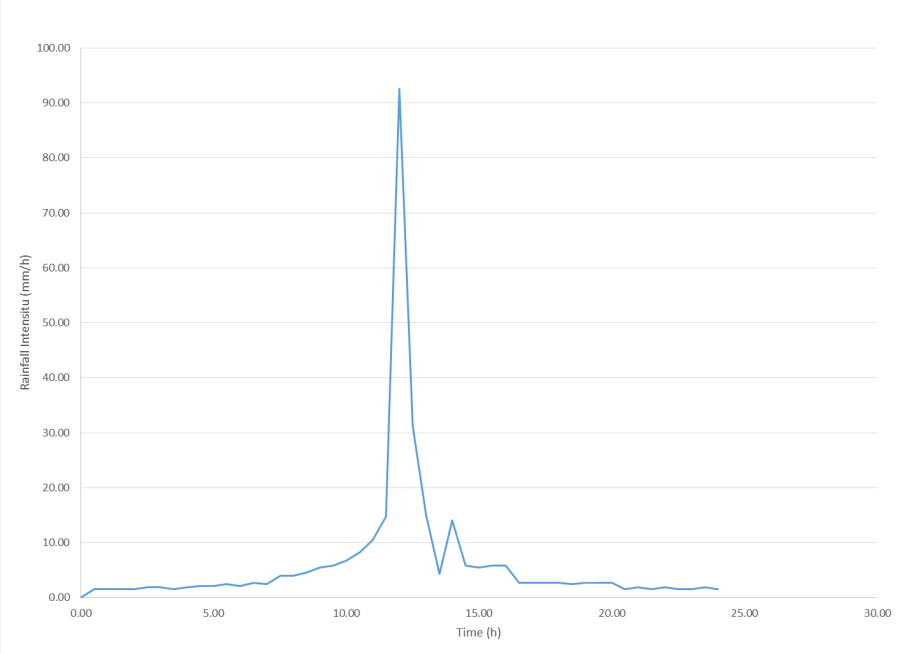


Figure 7.3 – Rainfall intensity time series.

7.4 Create the Infiltration Parameters Data File

To account for infiltration you will create polygons where you specify the infiltration calculation method and its corresponding data. Each polygon may use a different method with its associated parameters. You will need to create the infiltration ASCII text files for each polygon. These files can be created with any text editor such as Notepad or Wordpad. The infiltration file is described in detail in the RiverFlow2D Reference Manual. For this tutorial we will use two infiltration files, `Infiltration1.dat` and `Infiltration2.dat`, provided in the folder for this tutorial.

7.5 Create the Rainfall/Evaporation Polygons

We will create the rainfall/evaporation polygons in the *RainEvap* area coverage.

Note that the *RainEvap* coverage name should not be changed. If you use a different name for this coverage, the model will not be able recognize the rainfall/evaporation polygons.

1. First click on the on the *RainEvap* coverage to make sure it is the active layer.
2. Using the *Create Feature Arc* tool, draw a polygon similar to the one shown in Figure 7.4.

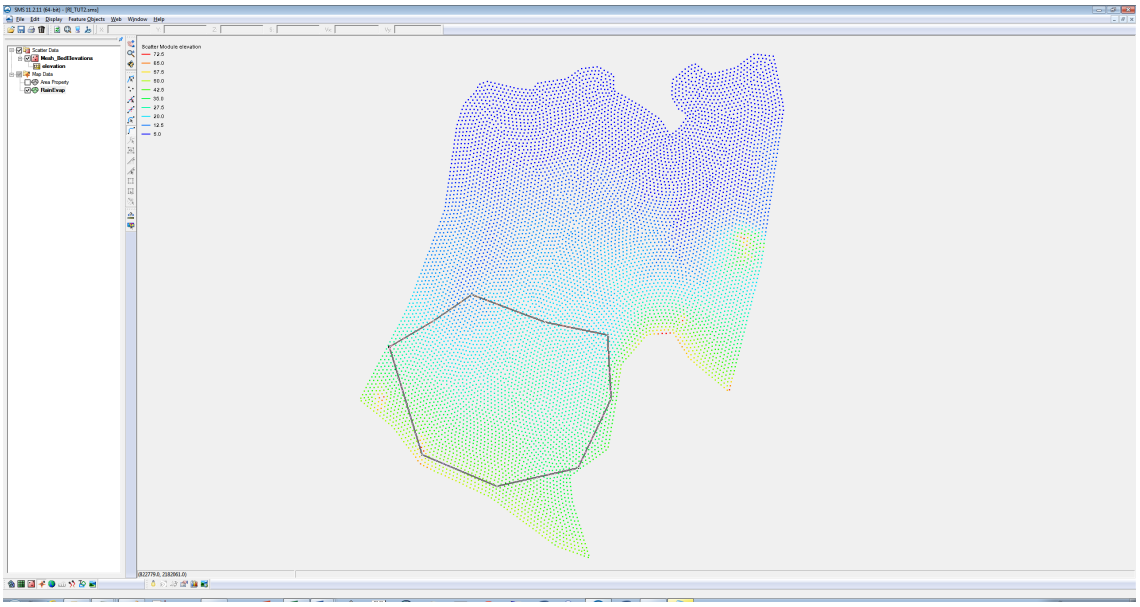


Figure 7.4 – Rainfall/Evaporation polygon.

If you are creating more than one polygon, make sure that they do not have common edges. If you need to have side-by-side polygons, create each one independently in such a way that even though they may have overlapping edges, the polygons are unique (see Figure 7.5).

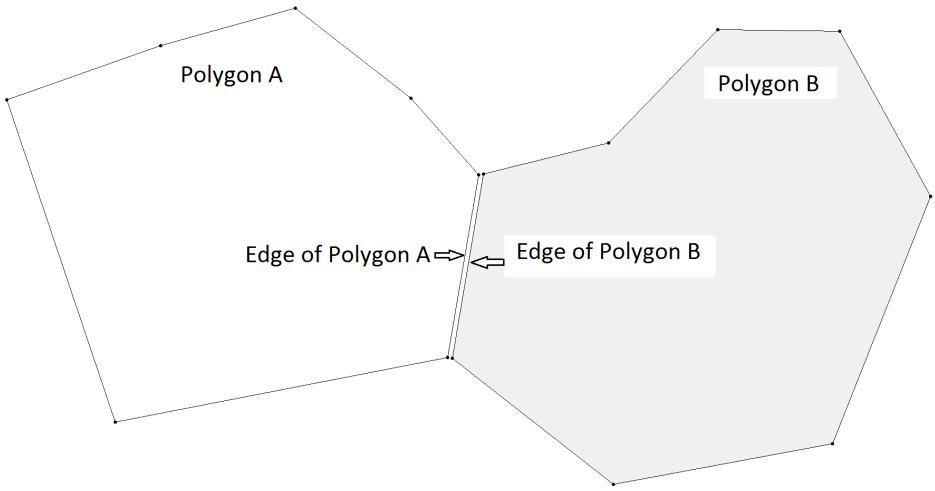


Figure 7.5 – Polygons must be independent.

3. On the *Feature Objects* menu, click *Build Polygons* to finalize the polygon.

   To generate the data structures correctly, the polygon needs to be defined by nodes instead of vertices. To enforce this, click on the *Select Feature Vertex* icon  , on the *Edit* menu, click *Select All*, then right-click on the plot window. On the pop-up menu, click *Convert to Nodes* to convert all vertices to nodes.

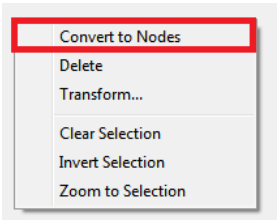


Figure 7.6 – Convert vertices to nodes.

4. Now, click the *Polygon Selection* icon  , click inside the polygon to select it, and right click. On the pop-up menu, click *Attributes*:

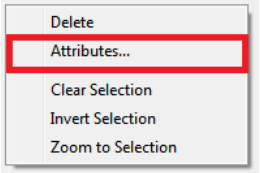


Figure 7.7 – Assign Rainfall/Infiltration data file name to polygon.

5. In the dialog select *Material*, `Rainfall1.dat` from the drop-down material list, and click OK:

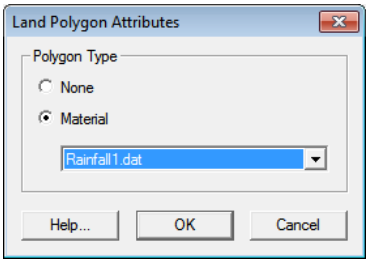


Figure 7.8 – Select rainfall data file name for polygon.

This completes the creation of rainfall/evaporation polygons.

7.6 Create the Infiltration Polygons

We will now create the infiltration polygons in the *Infiltration* area coverage.

Note that the coverage name: *Infiltration*, should not be changed. If you use a different name for this coverage, the model will not be able recognize the infiltration polygons.

1. To create the polygons, first click on the on the *Infiltration* coverage to make sure it is the active layer.

2. Then, using the *Create Feature Arc*, draw a polygon as shown:

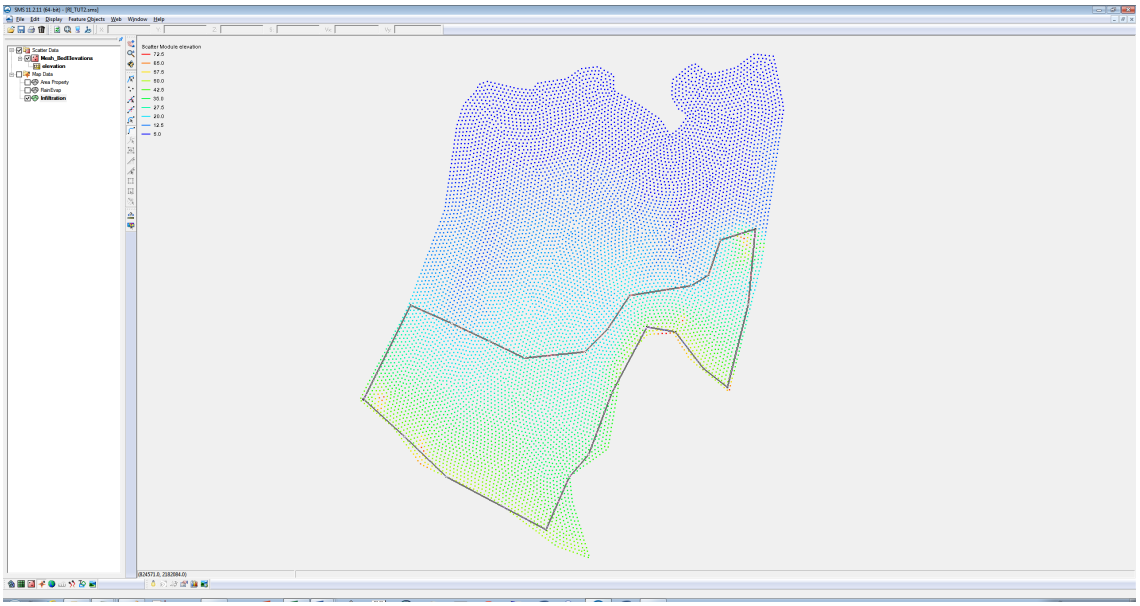


Figure 7.9 – Infiltration polygon.

3. On the *Feature Objects* menu, click *Build Polygons* to finalize the polygon.

To generate the data structures correctly, the polygon needs to be defined by nodes instead of vertices. To enforce this,

4. Click on the *Select Feature Vertex* icon  , on the *Edit* menu.
5. Click *Select All*, then right-click on the plot window and on the pop-up menu.
6. Click *Convert to Nodes* to convert all vertices to nodes.

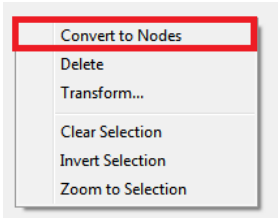


Figure 7.10 – Convert vertices to nodes.

7. Use the *Polygon Selection* icon  , click inside the polygon to select it, and right click. On the pop-up menu, click *Attributes*:

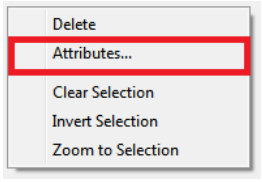


Figure 7.11 – Assign infiltration data file name to polygon.

8. In the dialog select *Material*, `Infiltration1.dat` from the drop-down material list, and click OK:

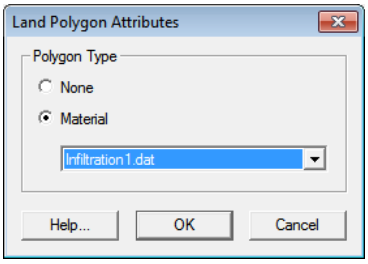


Figure 7.12 – Select infiltration data file name for polygon.

9. Now click on the *Infiltration* coverage again and using the *Create Feature Arc* draw a second polygon as shown:

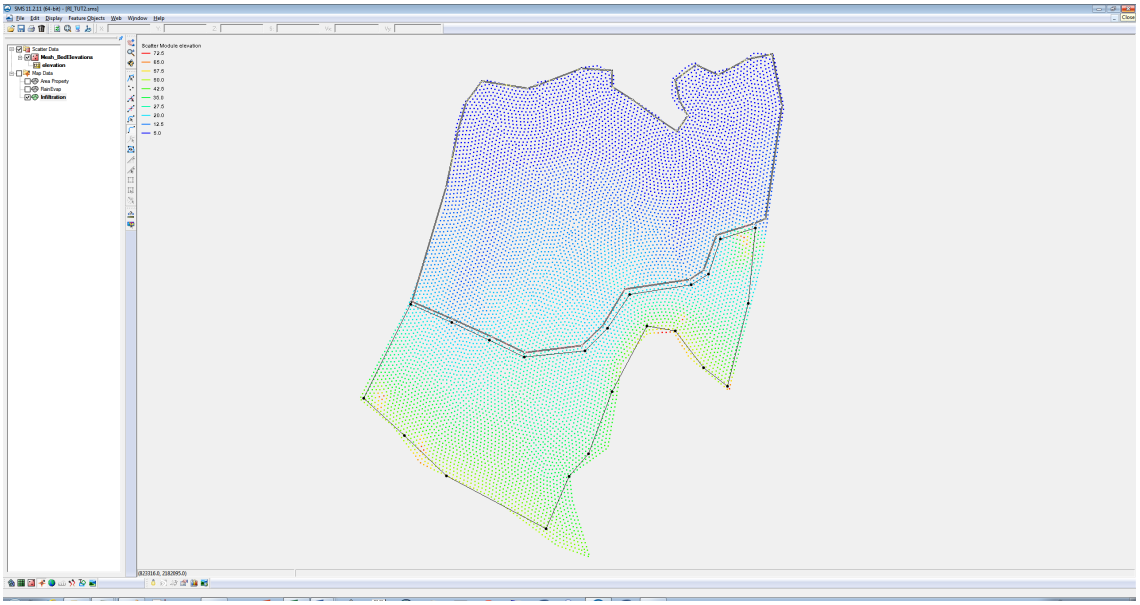


Figure 7.13 – Second infiltration polygon.

> Make sure that the polygons you draw do not overlap. Overlapping polygons create intersected areas that will confuse the algorithm used to define the infiltration and rainfall/evaporation areas.

10. Repeat the procedure as explained before to assign the `Infiltration2.dat` to the second polygon.

This completes the creation of the infiltration polygons.

7.7 Generate the Mesh and Copy Rainfall and Infiltration Files to the Mesh Folder

To finalize the spatial data preparation, click on the *Mesh* coverage under *Map Data*, and use the *Map* → *2D Mesh* on the Feature Object menu to generate the mesh. Then copy `Rainfall1.dat`, `Infiltration1.dat` and `Infiltration2.dat` to the mesh folder. In this tutorial this folder should be
...\ExampleProjects\RainfallInfiltrationTutorial\RI_TUT2\RiverFlow2D\Mesh.

> Note that when SMS creates a mesh it assigns to it a name under the Mesh data coverage group as shown in Figure 10.8. The project folder always has the same name as the mesh name.

7.8 Running the Model

1. Click on the *Mesh Data* Coverage,

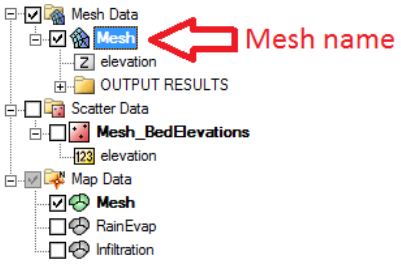


Figure 7.14 – The project folder and the mesh share the same name.

2. Click *Run* ***RiverFlow2D*** on the RiverFlow2D menu. The Data Input Program will appear.

3. Select Rainfall and Infiltration on the *Control Data* panel, set the simulation time to 12 hours as shown in Figure 7.15.

4. Save the project.

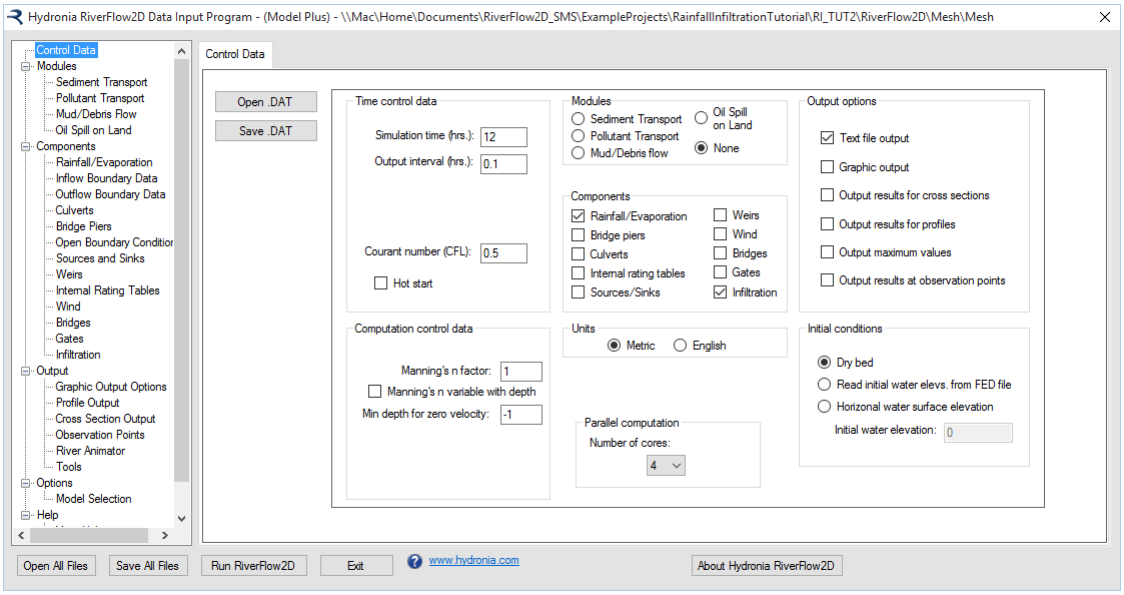


Figure 7.15 – Control Data panel.

5. Finally, click the *Run RiverFlow2D* button. Figure 7.16 shows the dynamic window at the end of the run.

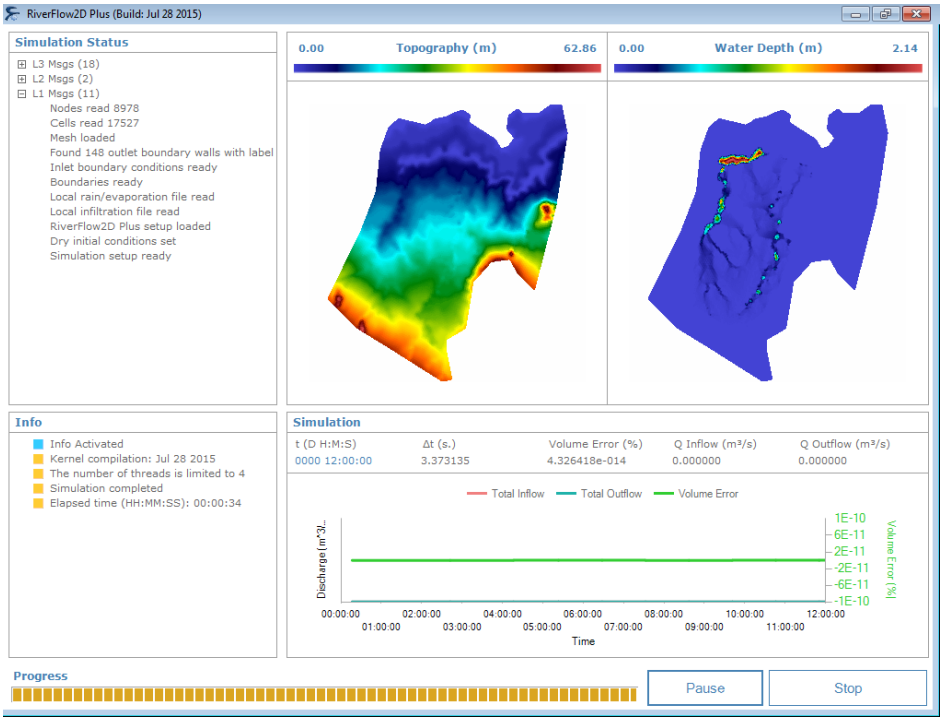


Figure 7.16 – RiverFlow2D window at the end of the Rainfall/Infiltration tutorial run.

6. Click *Stop* and close the window using the *Close Window* icon on the upper right of the window.

7.9 Visualizing Rainfall/Infiltration Results

When the model completes the execution and has generated all of result files, you can use SMS visualization tools to explore results. This is accomplished by opening the `Mesh_RESULTS.HDF5` file that the model creates in the project folder. In addition to the usual plots (depths, velocities, etc.) in this case the graphic options include the Accumulated Infiltration which is the total accumulated infiltration volume into the soil and the Accumulated Rainfall that represents the total volume from precipitation at each cell (see Figure 7.17).

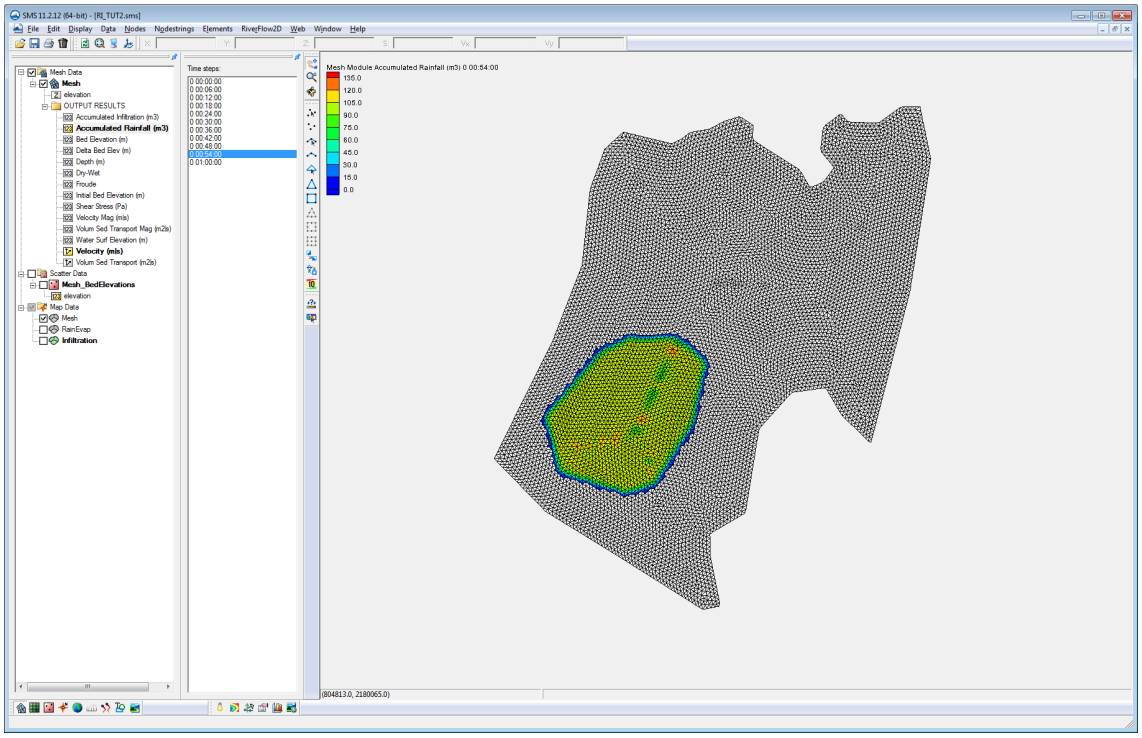


Figure 7.17 – Visualization of rainfall and infiltration results.

8 Setting up Initial Water Surface Elevations

For applications where the initial water surface elevation needs to be specified over some areas such as reservoirs, and other water bodies you can define set the initial water elevation over any polygon.

This tutorial illustrates how to perform a simulation with an area where the water elevation is set to a specified value. The procedure includes the following steps:

1. Create new values (Materials) corresponding to the initial water elevations.
2. Open an existing RiverFlow2D project.
3. Create a *InitialConditions* coverage and the polygons defining the area.
4. Generate the mesh.
5. Run the model.

The files required to follow this tutorial can be found in the following directory:
`...\Documents\RiverFlow2D_SMS\ExampleProjects\IntialConditionTutorial`

8.1 Open an Existing Project

On the *File* menu, click *Open* to open the existing project: `InitialConditions.sms`.

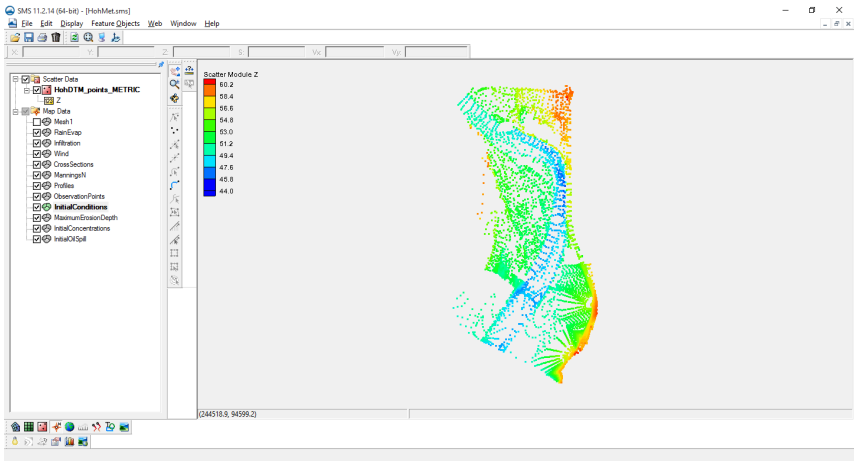


Figure 8.1 – Existing RiverFlow2D project.

This project already contains a terrain elevation dataset, a model domain extent polygon, outflow boundary conditions, and empty *InitialConditions* coverage.

If you are starting a new project that does not include the Initial Condition Coverage you may create one as follows:

1. Right-click on *Map Data* and click on *New Coverage*
2. On the dialog select *Area Property*
3. Enter *InitialConditions* as *Coverage Name*
4. Click OK

8.2 Create New Initial Water Surface Elevation Value

Use the *Material Data Command* in the *Edit* menu to create new *Material Data* entries with numeric values that will correspond with the initial water elevation that you want to associate with each polygons. To add new values, use the *New* button, and enter Material values. In this case we will enter `60.0`:

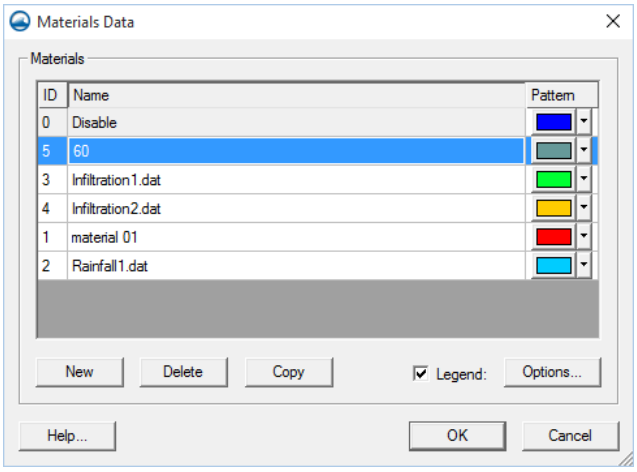


Figure 8.2 – Materials Data Dialog to enter initial water elevation.

8.3 Create the Initial Water Elevation Polygon

We will create an initial water elevation polygon in the *InitialConditions* area coverage.

Note that the *InitialConditions* coverage name should not be changed. If you use a different name for this coverage, the model will not be able recognize the polygons.

To create the polygon, first click on the on the *InitialConditions* coverage to make sure it is the active layer, and using the *Create Feature Arc* tool, draw a polygon as shown:

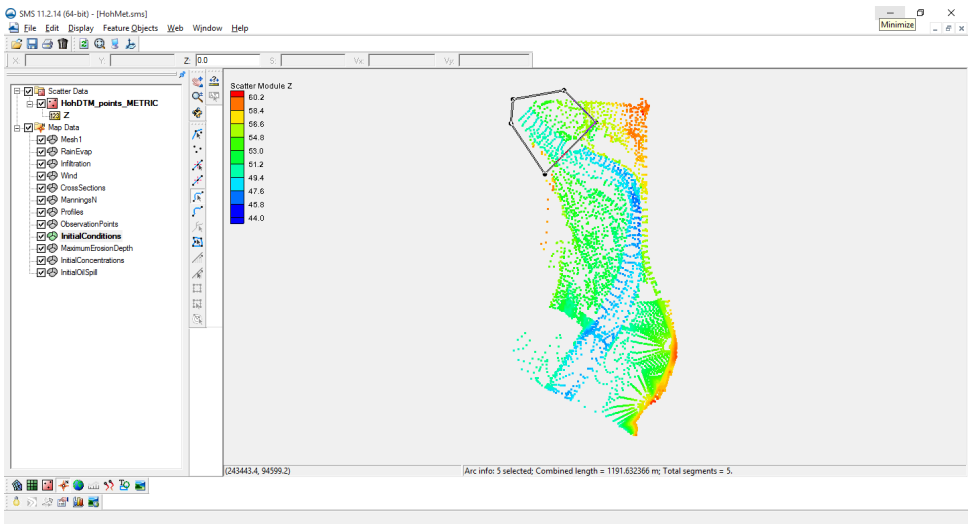


Figure 8.3 – Initial water elevation polygon.

On the *Feature Objects* menu, click *Build Polygons* to finalize the polygon.

To generate the data structures correctly, the polygon needs to be defined by nodes instead of vertices. To enforce this, click on the *Select Feature Vertex* icon  , on the *Edit* menu, click

Select All, then right-click on the plot window and on the pop-up menu, click *Convert to Nodes* to convert all vertices to nodes.

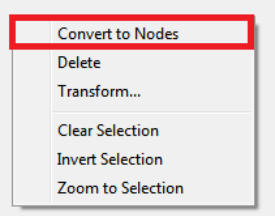


Figure 8.4 – Convert vertices to nodes.

If you are entering more than one initial conditions polygon, make sure that they do not have common edges. If you need to have side-by-side polygons, create each one independently in such a way that even though they may have overlapping edges, the polygons are unique (see Figure 8.5).

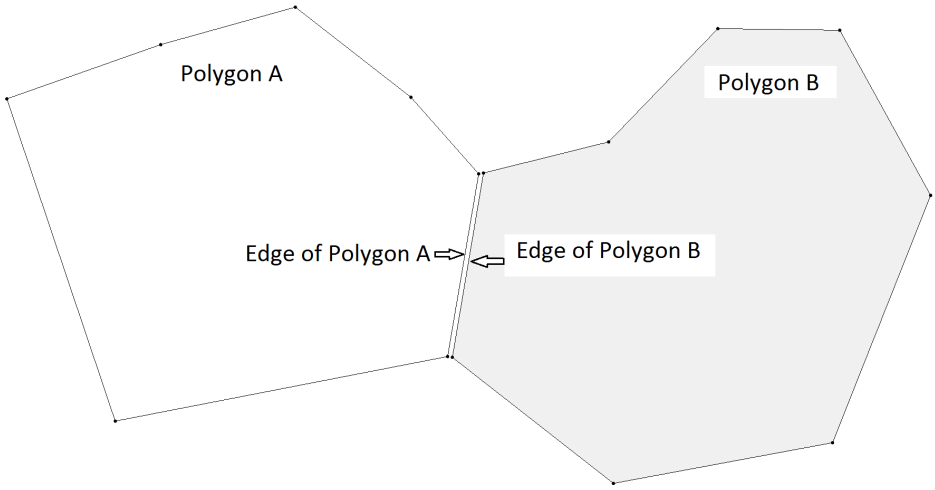


Figure 8.5 – Polygons must be independent.

Use the *Polygon Selection* icon , click inside the polygon to select it, and right click. On the pop-up menu, click *Attributes*:

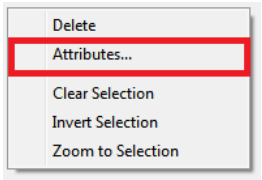


Figure 8.6 – Assign a water elevation value to polygon.

In the dialog select *Material*, 60 from the drop-down material list, and click OK:

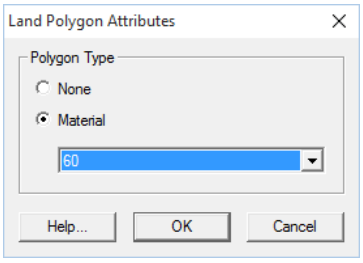


Figure 8.7 – Select value for polygon.

This completes the creation of the water elevation polygons.

8.4 Generate the Mesh

To finalize the spatial data preparation, click on the *Mesh* coverage under *Map Data*, and use the *Map* → *2D Mesh* on the *Feature Object* menu to generate the mesh.



Figure 8.8 – The project folder and the mesh share the same name.

8.5 Running the Model

To run the model do as follows:

1. Select the *Mesh Data* in the Project explorer to make the *Mesh Module* the active module.
2. On the *RiverFlow2D Menu* menu, click *Run RiverFlow2D*

   This will bring up the RiverFlow2D Data Input Program in which you can set all the run control parameters for RiverFlow2D.
3. Set the simulation time to 2 hours
4. Select the *Read initial water elevations from .FED file* in the *Initial conditions* frame

   Leave all other parameters at their default values.

Finally, click the *Run RiverFlow2D* button on the lower section of the Data Input Program. When the model finishes the execution, you can use the visualization tools to explore results.

This concludes the Setting Initial Water Surface Elevations Condition tutorial.

9 Using Manning's n Shapefiles

This tutorial illustrates how to use Manning's n shapefiles to assign n values to an existing RiverFlow2D project. The procedure includes the following steps:

1. Open an existing RiverFlow2D project.
2. Create a new Area Property coverage for Manning's n.
3. Read in the shapefiles with Manning's n polygons.
4. Map the shapefile to the Manning's n coverage.
5. Map the Manning's n coverage to the existing mesh.
6. Display the Manning's n on the existing mesh.
7. Enter n-values for each material.
8. Save the project and check that Manning's n have been updated in the .2dm file.

The files required to follow this tutorial can be found in the following directory:
`...\Documents\RiverFlow2D_SMS\ExampleProjects\ManningsNShapefileTutorial`

Files for a completed tutorial are also included in the following folder:
`...\Documents\RiverFlow2D_SMS\ExampleProjects\ManningsNShapefileTutorialFinal`

9.1 Open an Existing RiverFlow2D Project

In this example we will use the RiverFlow2D project included in the `ManningsNShapefileTutorial` folder.

1. Open SMS
2. On the *File* menu, click *Open*. Find the file `ManningsNShapefile.sms` in the data files folder for this tutorial. Select *Open*.

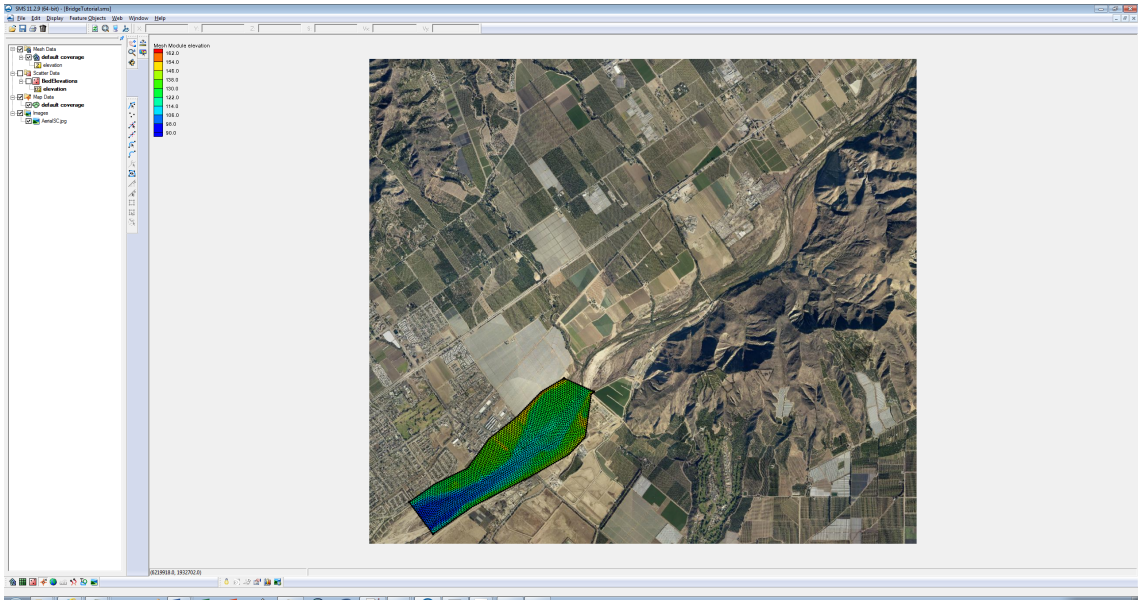


Figure 9.1 – RiverFlow2D project.

9.2 Create a New Area Property Coverage for Manning's *n*

In order to incorporate data into the RiverFlow2D model, the GIS data needs to be converted into native SMS data. Generally, GIS data is converted into Map coverages which are similar to GIS layers. Data converted from GIS layers into Map coverages are put into the active coverage. Therefore, it is important to have the type of coverage you need and make it active before the conversion process.

We will convert the Manning's n data into an area property coverage with designated materials (n-values).

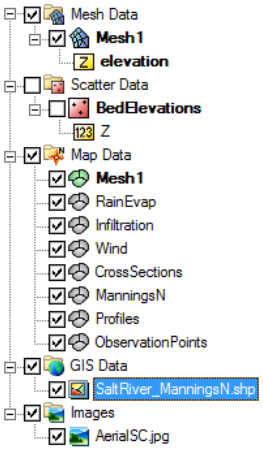
If in you project there is a *ManningsN* coverage, you do not need to create a new one. Otherwise, do as follows:

1. Right-click on the *Map data* layer group and select *New Coverage*.
2. In the *New Coverage* dialog select *Area Property* and enter the *Coverage name*: *ManningsN*. Click OK.

9.3 Read in the Shapefiles with Manning's *n* Polygons

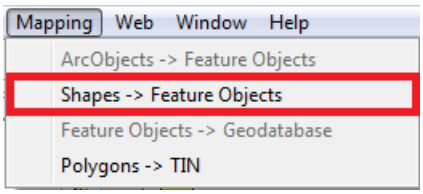
1. On the *File* menu, click *Open*. Find the file `SaltRiver_ManningsN.shp` in the data files folder for this tutorial. Select Open. There are three files with this name. The shapefile with

extension `.shp` is the one that should be read in.



9.4 Map the Shapefile to the Manning's *n* Coverage

1. In the GIS module (shapefile) and with ManningsN coverage active, click *Mapping* and then *Shapes* → *Feature Objects*. Click *Yes* to use all the shapes in the visible shapefiles for mapping.



If you have selected any number of shapes in the shapefile, SMS will not prompt you to use all shapes within visible shapefiles. Instead, it will only convert those selected shapes into feature objects. This is useful if you only plan on using a few selected shapes from a large shapefile.

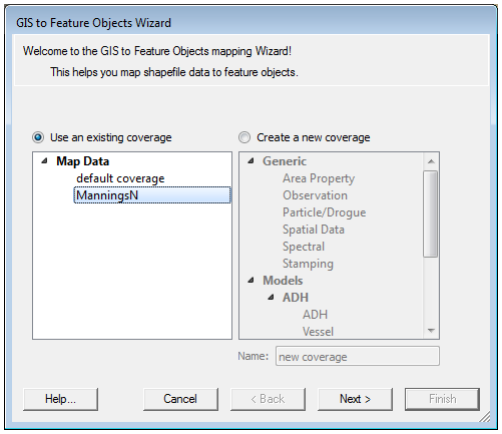


Figure 9.2 – GIS to Feature Objects Wizard dialog.

2. Select the *ManningsN* coverage and click *Next* to bring up the *GIS Feature Objects Wizard* table.

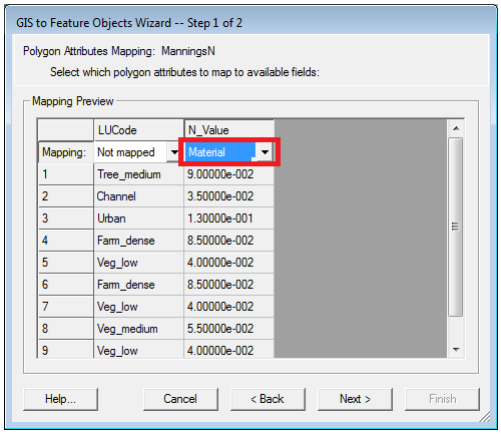


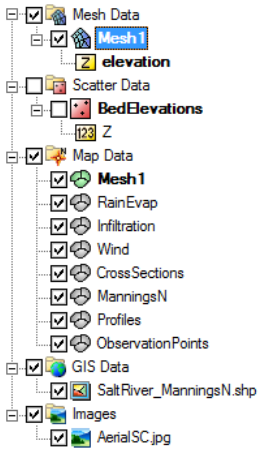
Figure 9.3 – Mapping polygons to materials.

3. In the Description column *N_Value*, select *Material* from the Mapping drop-down list. SMS will assign the polygon description to each polygon mapped to the active coverage.
4. Click *Next* then *Finish*.

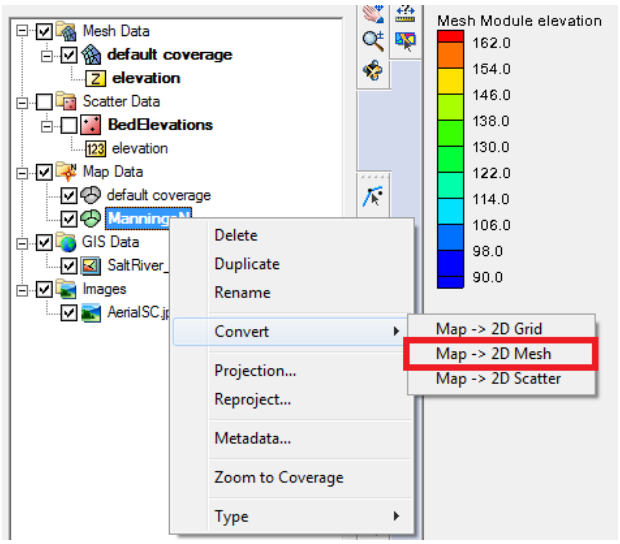
9.5 Map the Manning's *n* Coverage to the Existing Mesh

To assign the Manning's n to elements in a mesh do as follows:

1. Select the mesh you want to assign (since there can be more than one). In our case select as follows:



2. Right-click on the ManningsN area property coverage you want to use as a source and on the pop-up menu, click *Convert* and then *Map → 2D Mesh*. Since this is an area property coverage, it will not create elements, it will only assign material types to the elements.



9.6 Display the Manning's *n* on the Existing Mesh

The Manning's n data has now been copied to an area property coverage and assigned to mesh elements. To display the materials on the mesh do as follows:

1. Open the Display Options dialog clicking *Display Options* on the *Display* menu.
2. Select *2D mesh* in the tree menu to the left of the dialog.
3. Unselect Contours.
4. Select *Materials* and *Materials Numbers*.

5. Click *OK*

The display will look similar to the one shown on the following Figure.

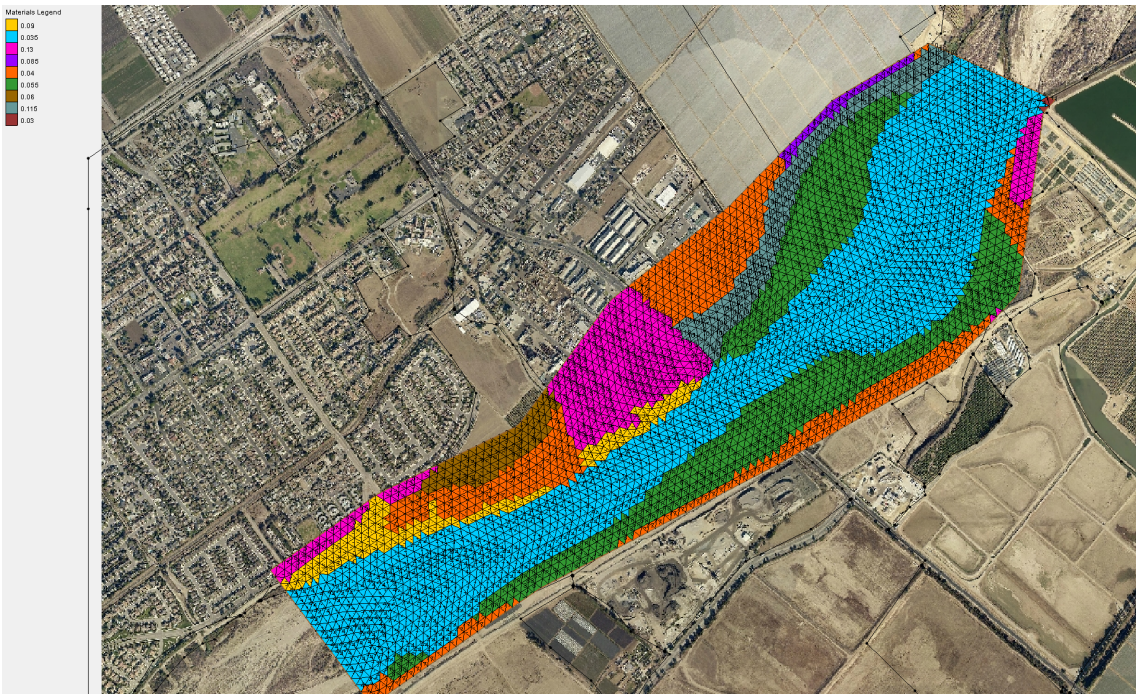


Figure 9.4 – Display of materials (Manning's n).

9.7 Enter n-values for Each Material

To complete the process you must enter the n-values for each material.

1. Select the *Mesh* module.
2. On the RiverFlow2D menu, click *Material Properties...*
3. In the *Material Properties* dialog enter the *Manning's n* values for each material as shown.

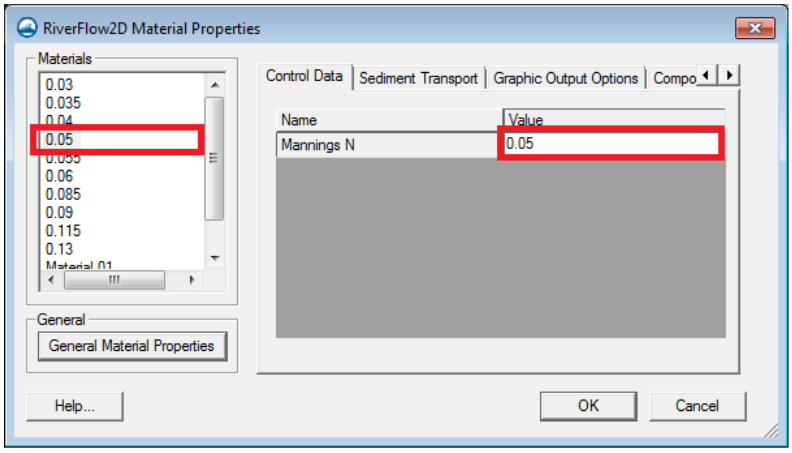
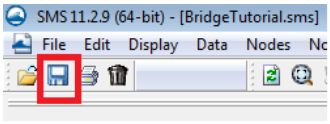


Figure 9.5 – RiverFlow2D Material properties.

4. Click OK to end the Manning's n assignment.

9.8 Save the Project and Check that Manning's n have been Updated in the .2dm file

1. Save the project using the save project icon:



2. Open the `mesh1.2dm` file using Notepad or any text editor. The file is in the following folder:

   `...\Documents\RiverFlow2D_SMS\ExampleProjects\ManningsNShapefileTutorial\RiverFlow2D\Mesh1\`

3. Go to the final part of the file and look for the MAT_VAL entries. They will look like these:

   MAT 11 "0.03"
   MAT 4 "0.035"
   MAT 7 "0.04"
   MAT 12 "0.05"
   MAT 8 "0.055"
   MAT 9 "0.06"
   MAT 6 "0.085"
   MAT 3 "0.09"
   MAT 10 "0.115"
   MAT 5 "0.13"
   MAT 1 "Material 01"
   MAT 2 "Material 02"
   MAT_MULTI 0

```
MAT_DEF 1 1 "Mannings N" 2 3.50000000e-002 0.00000000e+000 1.79769313e+308
ENDPARAMDEF
BEG2DMBC
GP_VAL 1 7 "Transient"
GP_VAL 3 2 -1
GP_VAL 3 3 -1
MAT_VAL 11 1 1 3.50000000e-002
MAT_VAL 4 1 1 3.50000000e-002
MAT_VAL 7 1 1 3.50000000e-002
MAT_VAL 12 1 1 5.00000000e-002
MAT_VAL 8 1 1 3.50000000e-002
MAT_VAL 9 1 1 3.50000000e-002
MAT_VAL 6 1 1 3.50000000e-002
MAT_VAL 3 1 1 3.50000000e-002
MAT_VAL 10 1 1 3.50000000e-002
MAT_VAL 5 1 1 3.50000000e-002
MAT_VAL 1 1 1 3.50000000e-002
MAT_VAL 2 1 1 3.50000000e-002
```

This concludes the Manning's n shapefile tutorial. You may continue to experiment with shapefiles in RiverFlow2D SMS by adding other pertinent GIS data into your projects.

10 Simulating Bed Load Sediment Transport with Limited Erosion Bed Areas

In the Sediment Transport model you can define areas with a maximum erosion depth. This is useful to represent pavements, rock outcrops or any surface that does not erode or that it has a known erodible layer of sediment above it.

This tutorial illustrates how to perform a sediment transport simulation in which there is a non-erodible area. The procedure includes the following steps:

1. Create new values (materials) corresponding to the maximum erosion depths.
2. Open an existing RiverFlow2D project.
3. Create a *MaximumErosionDepth* coverage and the polygons defining the limited erosion areas.
4. Generate the mesh.
5. Run the model.

The files required to follow this tutorial can be found in the following directory:
`...\Documents\RiverFlow2D_SMS\ExampleProjects\MaxErosionDepth`

10.1 Open an Existing Project

On the *File* menu, click *Open* to open the existing project: `MaxErosionDepth.sms`.

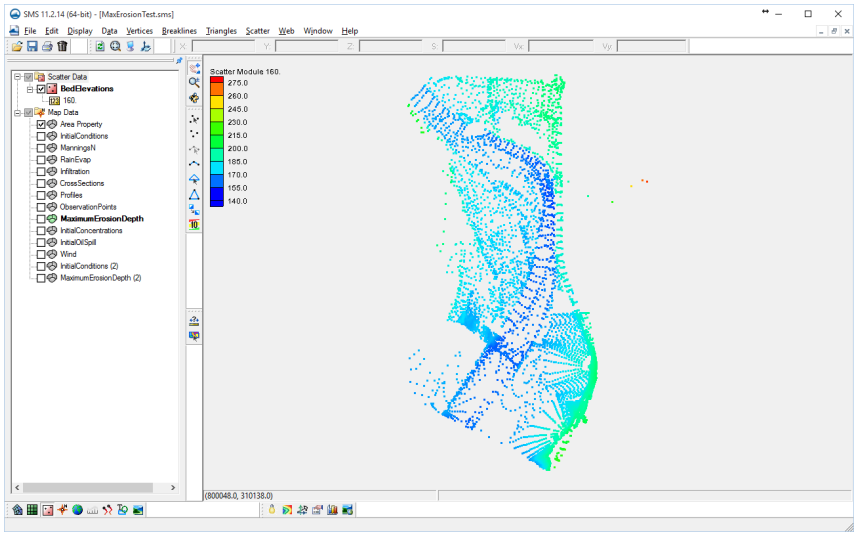


Figure 10.1 – Existing RiverFlow2D project.

This project already contains a terrain elevation dataset, a model domain extent polygon, outflow boundary conditions, and empty *MaximumErosionDepth* coverage.

If you are starting a new project that does not include the *MaximumErosionDepth* Coverage you may create one as follows:

1. Right-click on *Map Data* and click on *New Coverage*
2. On the dialog select *Area Property*
3. Enter *MaximumErosionDepth* as *Coverage Name*
4. Click OK.

10.2 Create New Maximum Erosion Depth Values

Use the *Material Data Command* in the *Edit* menu to create new *Material Data* entries with numeric values that will correspond tow the maximum erosion depth that you want to associate with each polygon. To add new values, use the *New* button, and enter Material values. In this case we will enter `0.0`:

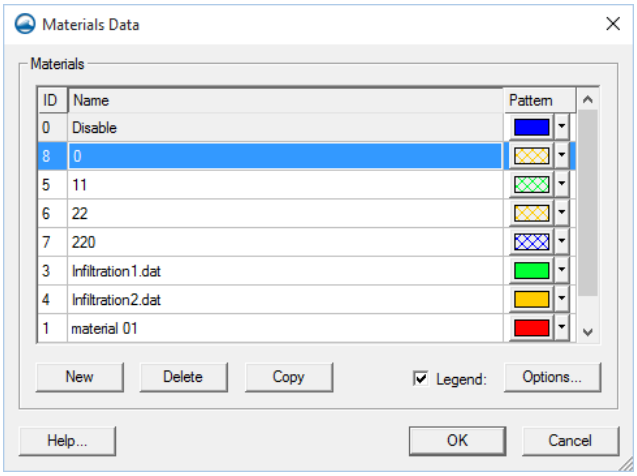


Figure 10.2 – Materials Data Dialog to enter maximum erosion depths.

10.3 Create the Maximum Erosion Depth Polygon

We will create a maximum erosion depth polygon in the *MaximumErosionDepth* area coverage.

Note that the *MaximumErosionDepth* coverage name should not be changed. If you use a different name for this coverage, the model will not be able recognize the polygons.

1. To create the polygon, first click on the on the *MaximumErosionDepth* coverage to make sure it is the active layer.
2. Using the *Create Feature Arc* tool, draw a polygon as shown:

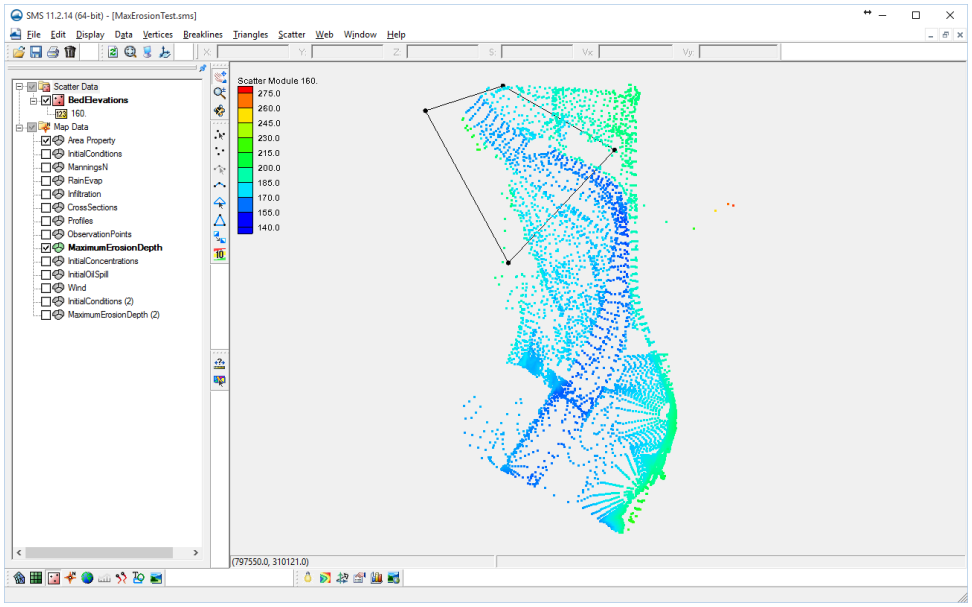


Figure 10.3 – Maximum erosion depth polygon.

3. On the *Feature Objects* menu, click *Build Polygons* to finalize the polygon.

If you are entering more than one polygon, make sure that they do not have common edges. If you need to have side-by-side polygons, create each one independently in such a way that even though they may have coincident edge, the polygons are unique (see Figure 10.4).

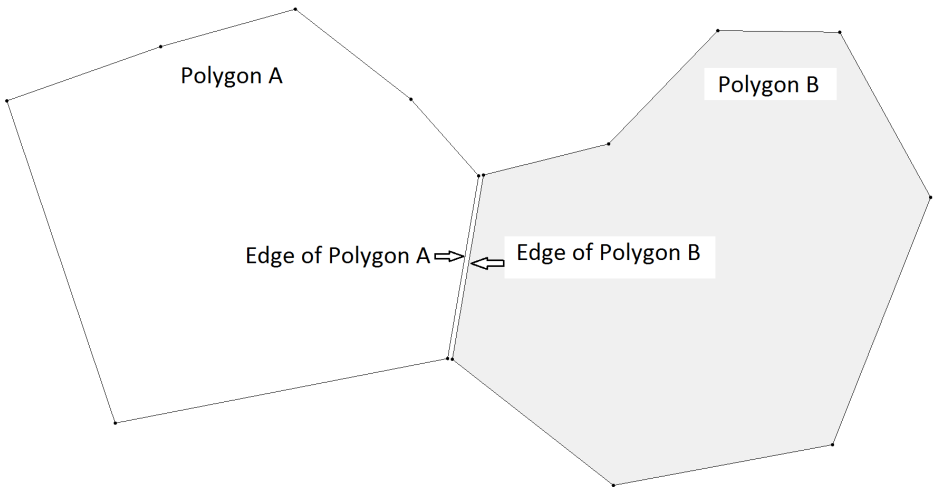


Figure 10.4 – Polygons must be independent.

4. To generate the data structures correctly, the polygon needs to be defined by nodes instead of vertices. To enforce this, click on the *Select Feature Vertex* icon , on the *Edit* menu, click *Select All*, then right-click on the plot window and on the pop-up menu, click *Convert to Nodes* to convert all vertices to nodes.

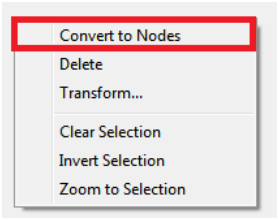


Figure 10.5 – Convert vertices to nodes.

5. Use the *Polygon Selection* icon , click inside the polygon to select it, and right click. On the pop-up menu, click *Attributes*:

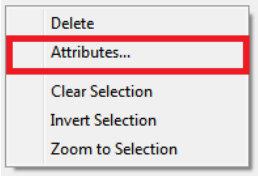


Figure 10.6 – Assign maximum erosion depth value to polygon.

6. In the dialog select *Material* 0 from the drop-down material list, and click OK:

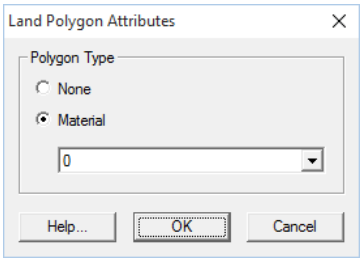


Figure 10.7 – Select value for polygon.

This completes the creation of maximum erosion depth polygons.

10.4 Generate the Mesh

To finalize the spatial data preparation, click on the *Mesh1* coverage under *Map Data*, and use the *Map* → *2D Mesh* on the Feature Object menu to generate the mesh.



Figure 10.8 – The project folder and the mesh share the same name.

10.5 Set the Global Parameters

1. Click on the *Mesh Data* Coverage, and on the the RiverFlow2D menu, click *Global Parameters*. Select the options in *Bed Load Sediment Transport* as indicated:

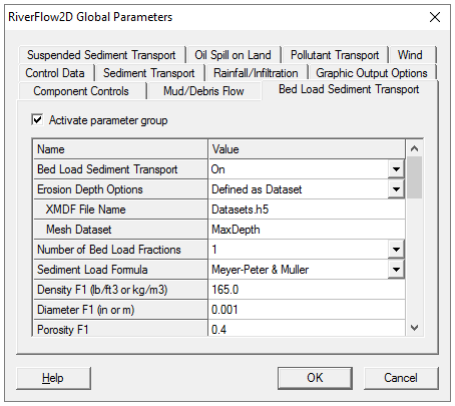


Figure 10.9 – Bed Load Sediment Transport tab.

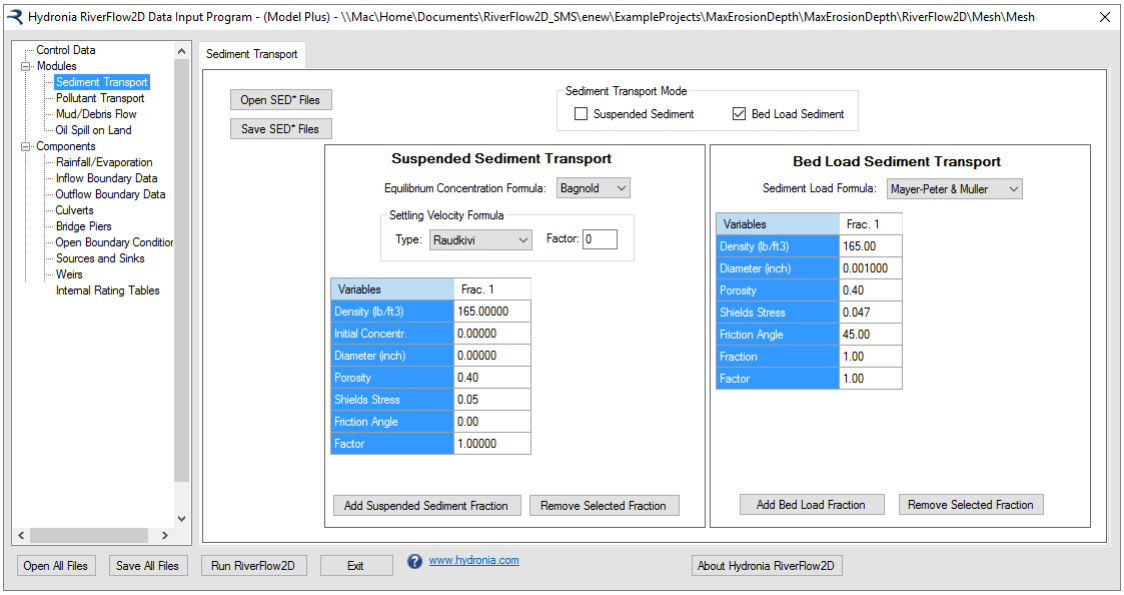
> Make sure to set the *Erosion Depth Options* to *Defined as Dataset* as shown.

2. Then, on the *Control Data* tab, set the simulation time to 12 hours, and save the project.

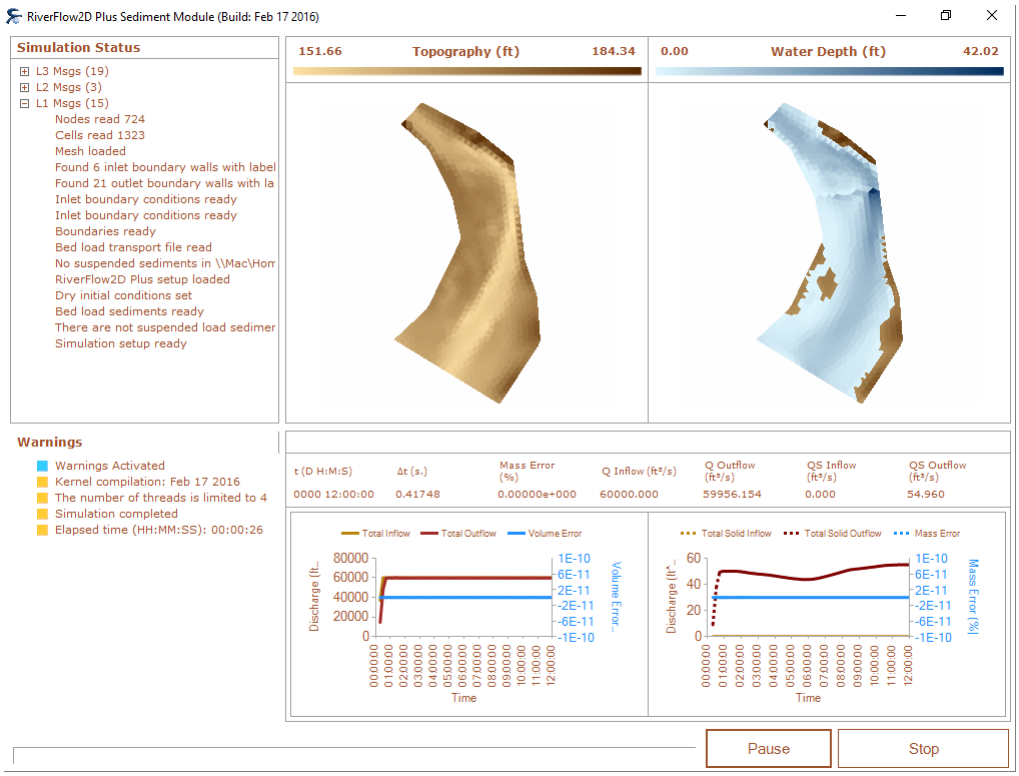
10.6 Running the Model

To run the model,

1. Select the *Mesh Data* in the Project explorer to make the *Mesh Module* the active module.
2. On the ***RiverFlow2D*** *Menu* menu, click *Run RiverFlow2D*. This will bring up the RiverFlow2D Data Input Program in which you can set all the run control parameters for the RiverFlow2D including the Bed Load Sediment Transport parameters in the *Sediment Transport* Panel:



3. Click the *Run RiverFlow2D* button on the lower section of the main DIP window to run the program. You will see a the dynamic window similar to this:



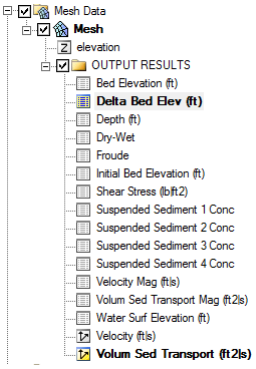
4. Close the dynamic window.

10.7 Post-processing the Model Results

To view the model results in graphic form, do as follows:

1. Click on *File* and click *Open*
2. On the dialog search the project folder where the results files are located. Typically, this folder is a few levels down the directory structure.
3. Click on the file *Mesh_RESULTS.HDF5* and then click *Open*. A new set of graphic layers will appear under the Mesh coverage:
4. Select *Delta Bed Elev (ft)* to view the erosion and deposition graph.
5. Click *Display Options* dialog.
6. On the 2D Mesh tab, select *Contours*.
7. On the *Contours* tab, select *Color Fill* from the *Contour method* drop-down list.
8. Click OK.

You can now click on the time step you wish to graph. The following figure shows a plot similar to the one you will see:



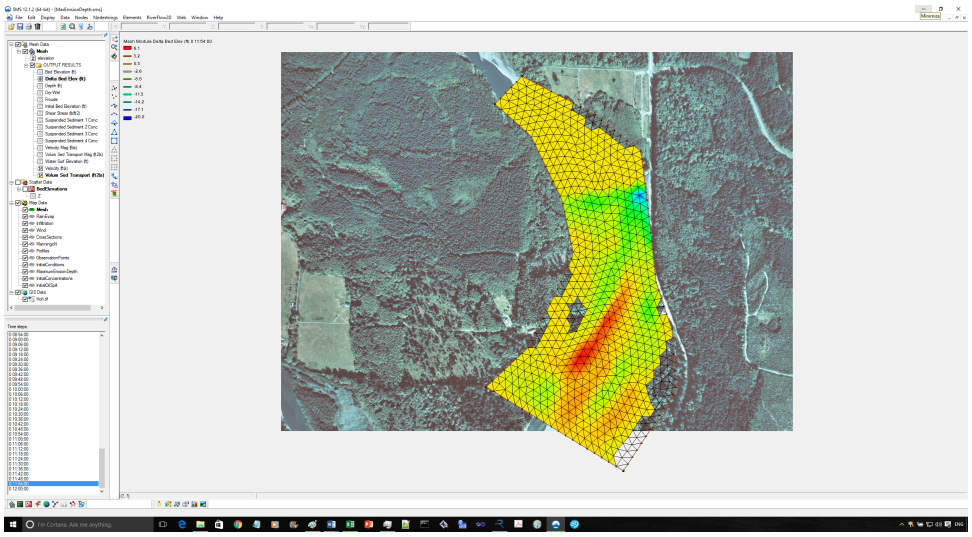


Figure 10.10 – Erosion and Deposition Graph.

Hovering the cursor over the non-erodible area will show a 0.0 erosion depth on the lower left part of the window.

11 Suspended Sediment Transport Simulations

This tutorial presents an application of the RiverFlow2D ST module to perform a suspended sediment transport simulation. It assumes that you have completed a simulation similar to the one described in the *Creating your First RiverFlow2D Project* tutorial presented earlier in this document. For convenience a completed simulation is included in the tutorial folder indicated below.
The following steps are discussed:

1. Open SMS
2. Open `SuspendedSedimentTutorial.sms`
3. Set suspended sediment data in the RiverFlow2D Data Input Program
4. Add the inflow concentrations for each suspended sediment fraction
5. Run RiverFlow2D from the Data Input Program
6. View RiverFlow2D Sediment Transport Model results

All the data files required to follow this tutorial are in the following directory:
`...\Documents\RiverFlow2D_SMS\ExampleProjects\SuspendedSedimentTutorial`

11.1 Run SMS and Open the Existing Project File

To open the project file do as follows:

1. Open the SMS Program double-clicking the SMS icon on your desktop  .
2. On the *File* menu, click *Open*.
3. Search for the Select the file `SuspendedSedimentTutorial.sms` in the folder for this tutorial and click the *Open* button.

11.2 Set suspended sediment data in the RiverFlow2D Data Input Program

To set the suspended sediment data, you first need to generate the `.2dm` file and create the RiverFlow2D data files. This is performed using the *RiverFlow2D* menu and clicking *Run RiverFlow2D*. A temporary window will appear and then the RiverFlow2D Data Input Program (DIP) window will display. Enter the data shown below making sure that the Sediment Transport module is activated:

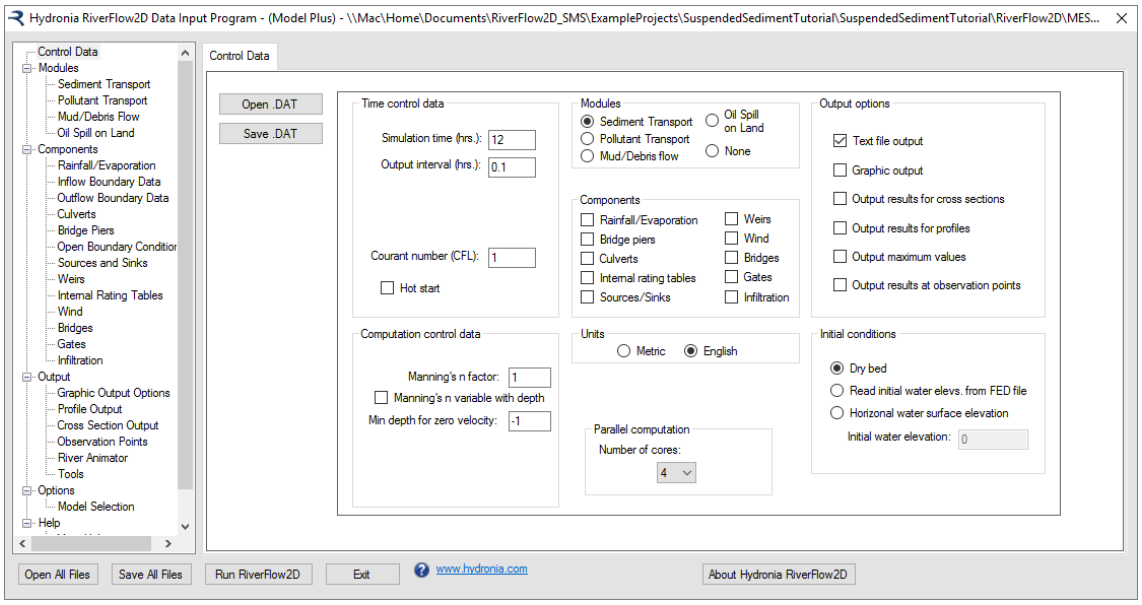


Figure 11.1 – RiverFlow2D Data Input Program with Sediment Transport Module activated.

In the Sediment Transport Panel, enter the suspended sediment data as shown. Note that the *Suspended Sediment* mode is selected and *Bed Load* is turned off.

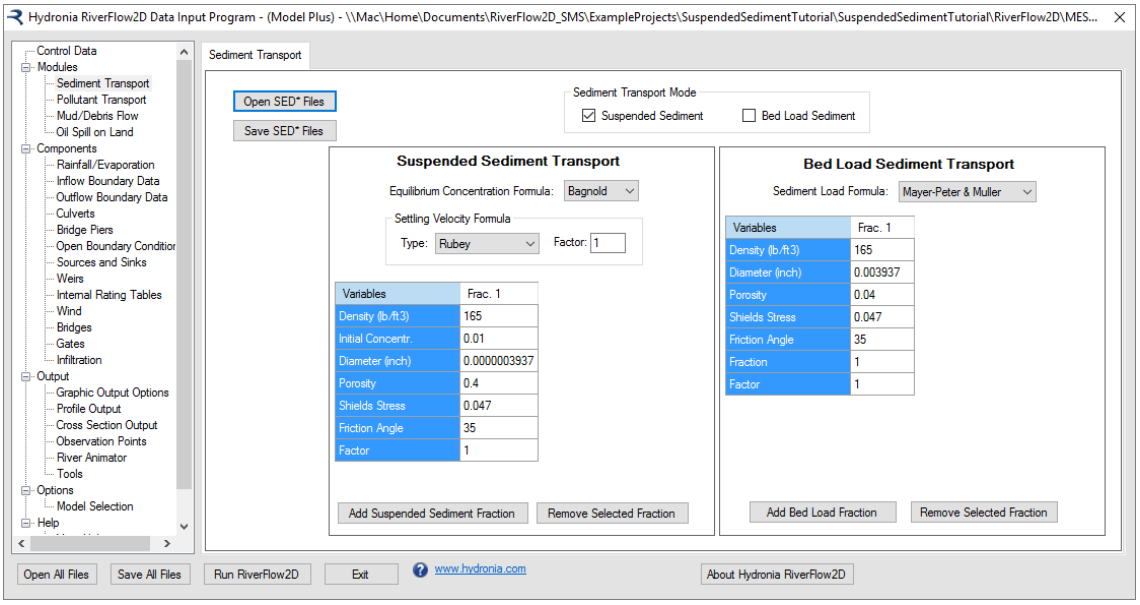


Figure 11.2 – RiverFlow2D Data Input Program with Suspended Sediment Data.

11.3 Add the Inflow Concentrations for each Fraction

When using the suspended sediment model, the user must impose volume concentration for each sediment fraction at all inflow boundaries. This data should be included in the same file that sets the discharge time series at each inlet. For instance, if you select one sediment fraction and BCType = 6 where the inflow is set as discharge, the inflow boundary condition file should be like this:

```
4
0.0          1000.          0.001
1.5          10000.         0.001
2.0          67000.         0.001
12.0         67000.         0.001
```

Where the first line indicates that there are 4 times. The first column corresponds to the time in hours: 0, 1.5, 2 and 12 hours in this example. The second column is the water discharge in ft^3/s. The third column indicate the sediment volume concentrations for the given fraction.

To enter this data first select the sediment transport panel. Only the inflow discharge will appear initially as shown:

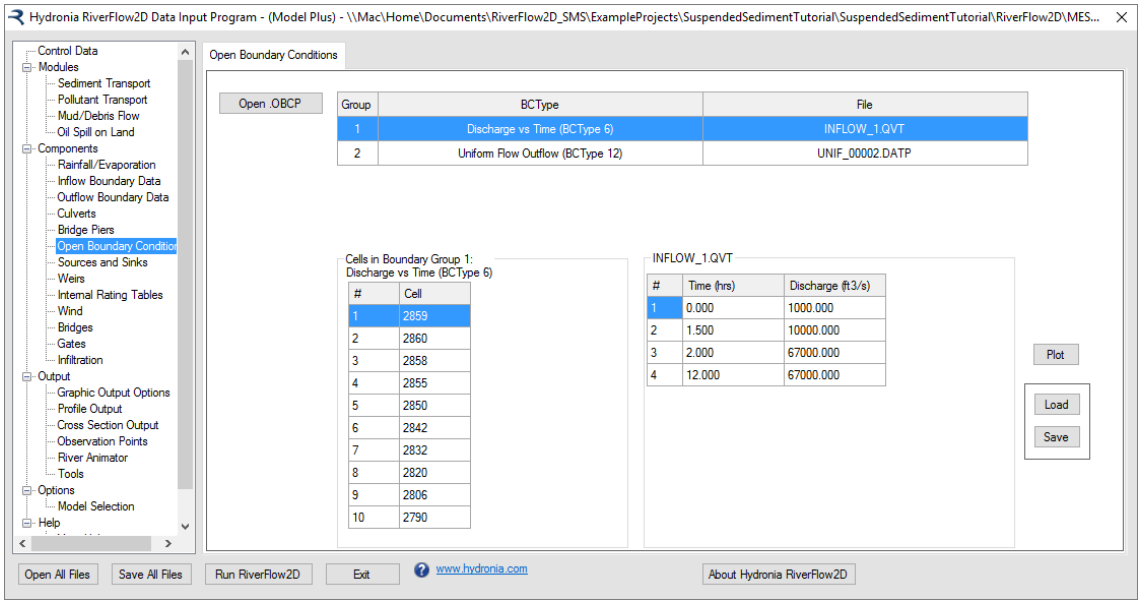


Figure 11.3 – RiverFlow2D Data Input Program showing the *Open Boundary Conditions* Panel.

Now click on the `INFLOW_1.QVT` file. A message box will remind you that you need to add one volumetric suspended sediment concentration for every fraction selected. You will need to enter the data for one column and all times as depicted:

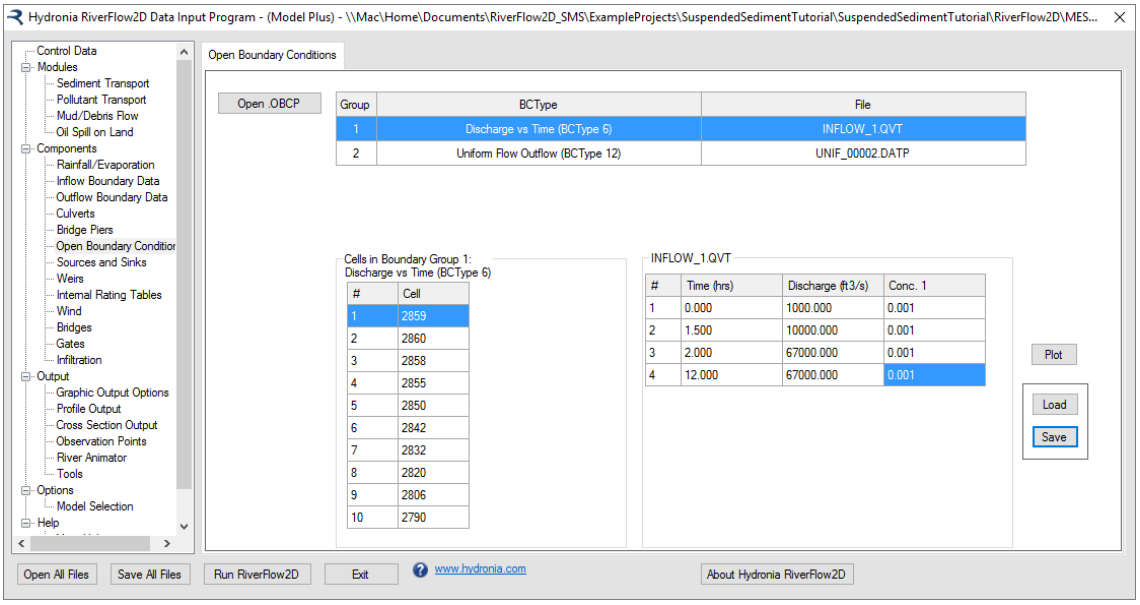


Figure 11.4 – RiverFlow2D Data Input Program showing the *Open Boundary Conditions* Panel with the completed concentration column.

Finally, click *Save* button under *Load*.

11.4 Run RiverFlow2D from the Data Input Program

To run the model do as follows:

1. On the lower frame of the RiverFlow2D Data Input Program, click *Run RiverFlow2D*.
2. In the following dialog, click *Yes* to save your changes.

The window shown in the figure below will appear indicating that the model has started running. The window also reports the simulation time, volume conservation errors for both water and sediment, total inflow and outflow discharges, and other parameters as the run progresses.

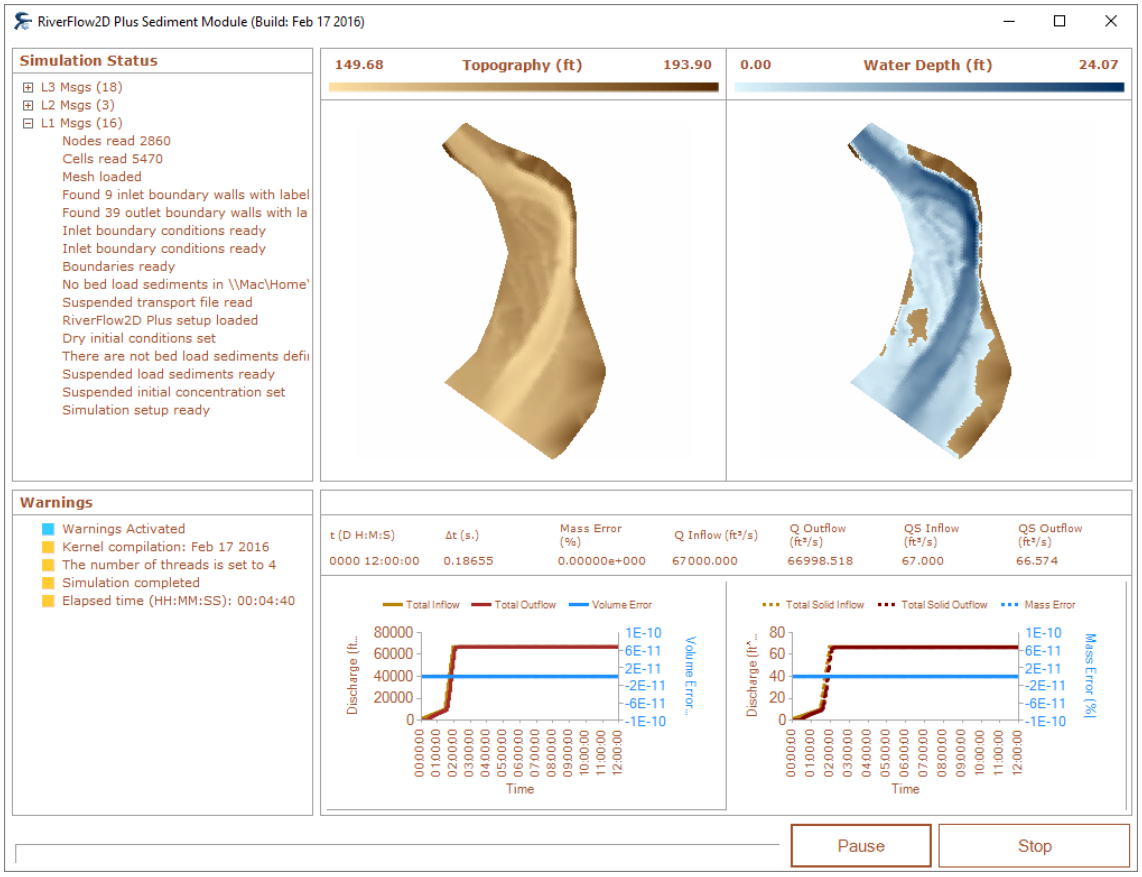


Figure 11.5 – Sediment Transport Model Run Report Window.

When the simulation finishes, close the dialog using the X icon on the upper right corner of the window. Another window will report on the final generation of post processing files.

11.5 View RiverFlow2D Sediment Transport Model Results

1. On the *File* menu, click *Open*.
2. Find the `MESH1_RESULTS.HDF5` file located in the folder where your project is saved and select the file. This folder will have the same name as the mesh (e.g. *MESH1*) and will be three directory levels below the one containing the `.sms` file.
3. Click the *Open* button.

Once the RiverFlow2D output file has been imported as Mesh Dataset you must decide on how to view the data. The Project Explorer may be used to select the desired Scalar and Vector output datasets. Scalar data sets are identified by the scalar icon and the vector datasets have the vector icon identifying them.

A convenient way to render the model results is to output the bed elevation changes editing the contour display options.

1. Activate the *Delta Bed Elev (ft)* output scalar dataset by selecting it in the Project Explorer.
2. Select the *Display Options* macro in the *Display Toolbar*.
3. In the *2D Mesh tab*, click the *All off button* to turn off current display options.
4. Turn on the *Contours*, and *Mesh boundary*.
5. Under the *Contours* tab, change the *Contour Method* to *Color Fill*.
6. Click OK to exit the dialog box
7. Select de desired time step.

SMS will redraw the screen similar to the one below.

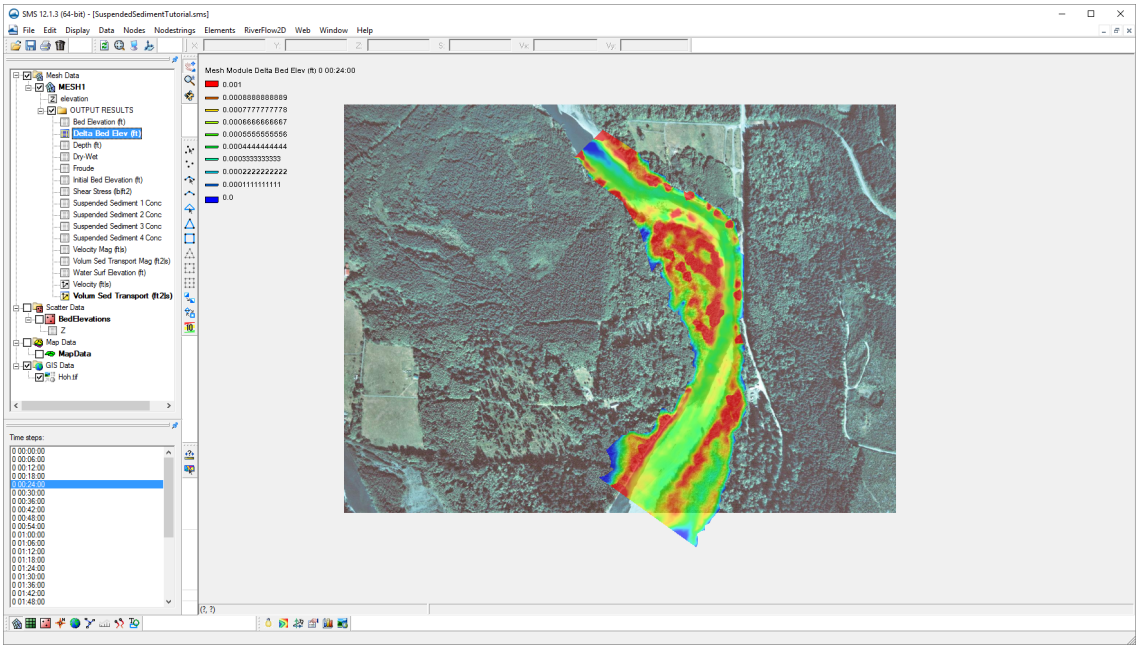


Figure 11.6 – Sediment Transport Model run result visualization.

This concludes the *Suspended Sediment Transport Simulations* tutorial.

12 Wind Driven Circulation

RiverFlow2D allows defining wind velocity on the water surface to account for the effect of the wind stress on the flow velocities. tThe conceptual model of a wind driven simulation with RiverFlow2D requires a series of *non-overlapping* polygons that determine the wind velocity data to the model. Only areas covered by polygons will be affected by the wind stress. Each wind velocity polygon should be associated with a file containing a wind velocity time-series file containing the two components of the wind velocity vector for each time. The user will need to generate the wind velocity data file associated with each polygon, and copy them to the project folder, prior to running the model.

This tutorial illustrates how to perform a wind drive simulation. The procedure includes the following steps:

1. Create new Materials corresponding to each wind velocity files.
2. Create wind velocity time-series data files.
3. Create the infiltration data file.
4. Open an existing RiverFlow2D project.
5. Create *Wind* coverage and the wind velocity polygons.
6. Generate the mesh.
7. Run the model.
8. Visualize model results.

The files required to follow this tutorial can be found in the following directory:
`...\Documents\RiverFlow2D_SMS\ExampleProjects\WindTutorial`

12.1 Open an Existing Project

On the *File* menu, click *Open* to open the existing project: `WindTutorial.sms`.

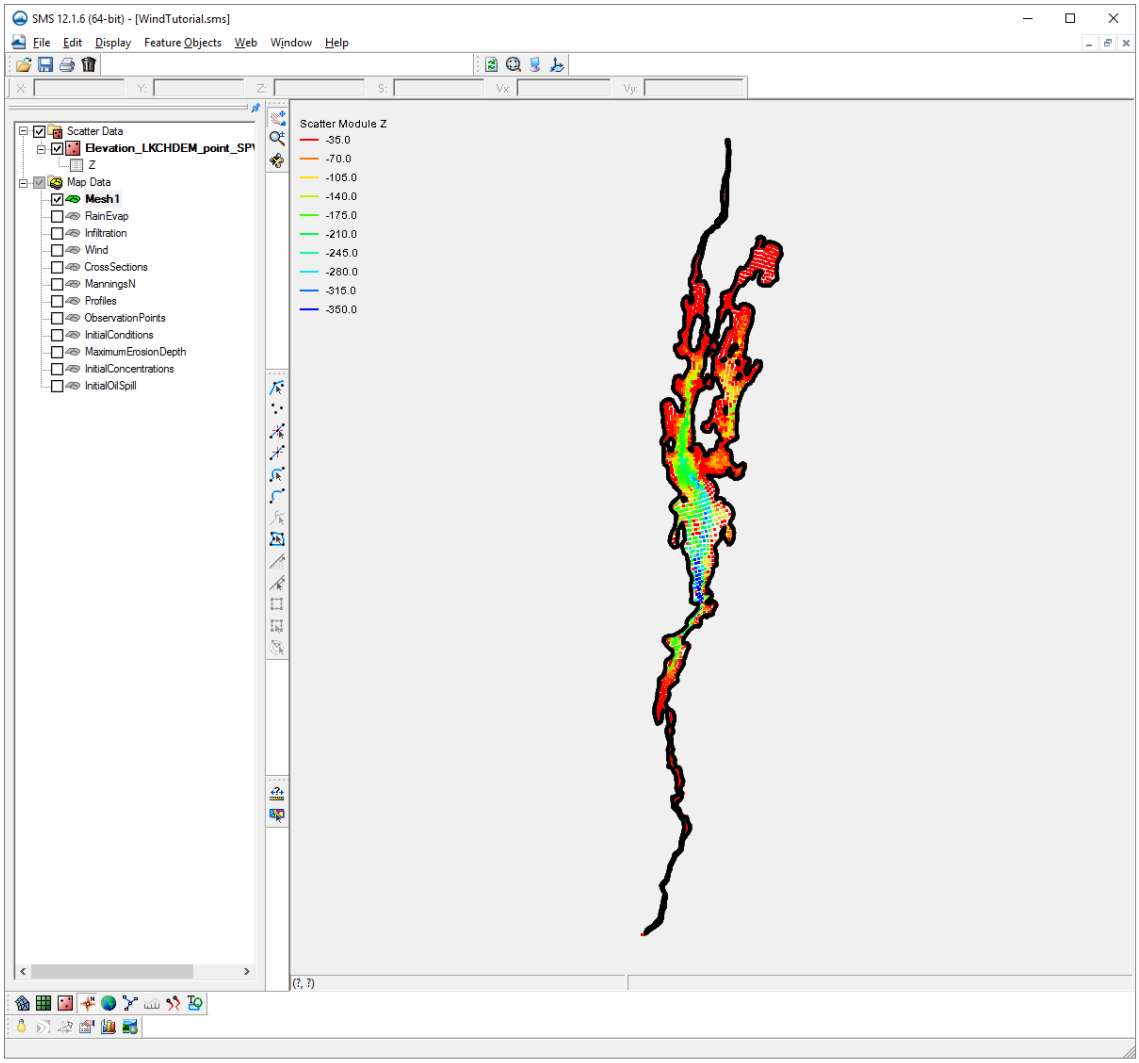


Figure 12.1 – Existing RiverFlow2D project.

This project represents a simulation of wind driven circulation in Lake Champlain in the USA, and already contains a bed elevation dataset, a model domain extent polygon, and an empty Wind area coverage.

12.2 Create New Materials Corresponding to Wind Velocity Files

1. Use the *Material Data* command in the *Edit* menu to create new *Material Data* entries. In the RiverFlow2D SMS GUI the wind velocity data file names will correspond to the material

names entered in the *Materials Data Dialog*.

2. To add new data file names, use the *New* button, and enter Material names that will correspond with the wind velocity data files that you want to associate with each polygon. In this tutorial we will enter `WindVelocDATA.DAT` as shown:

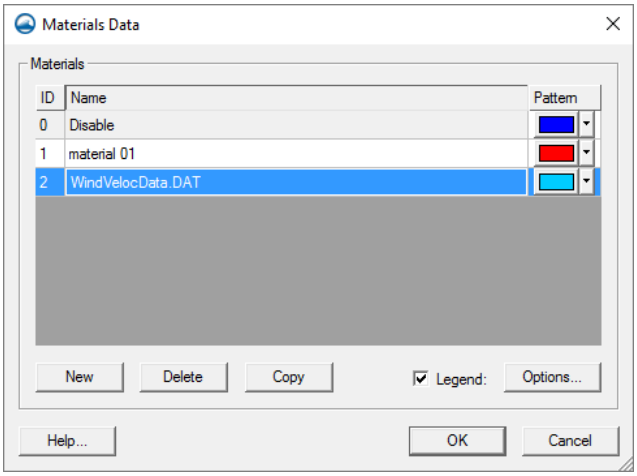


Figure 12.2 – Materials Data Dialog to enter wind velocity data file names.

12.3 Create the Wind Velocity Time-series Data File

To run a wind driven simulation with RiverFlow2D you will create polygons over which the wind velocity data will be applied, but first you need to create the wind velocity time series ASCII text files that will be associated with each polygon. These files can be created with any text editor such as Notepad or Wordpad. The wind velocity file has the following format:

Line 1: Number of points in the time series of wind velocity.

NP

NP lines containing:

TIME Wvx (m/s or ft/s) Wvy (m/s or ft/s)

where **Wvx** is the wind velocity vector component in the x direction, and **Wvy** is the wind velocity vector component in the y direction.

The following table is an excerpt of the `WindVelocDATA.DAT` file that is included in the Data folder for this tutorial.

```
6544
0          5.971128609          -2.165354331
1          5.085301837          8.825459318
2          3.838582677          6.627296588
3          5.872703412          4.921259843
...
```

12.4 Create the Wind Polygons

We will create one wind velocity polygon in the *Wind* area coverage.

Note that the *Wind* coverage name should not be changed. If you use a different name for this coverage, the model will not be able recognize the wind polygons.

1. First click on the on the *Wind* coverage to make sure it is the active layer.
2. Using the *Create Feature Arc* tool, draw a polygon similar to the one shown in Figure 12.3.

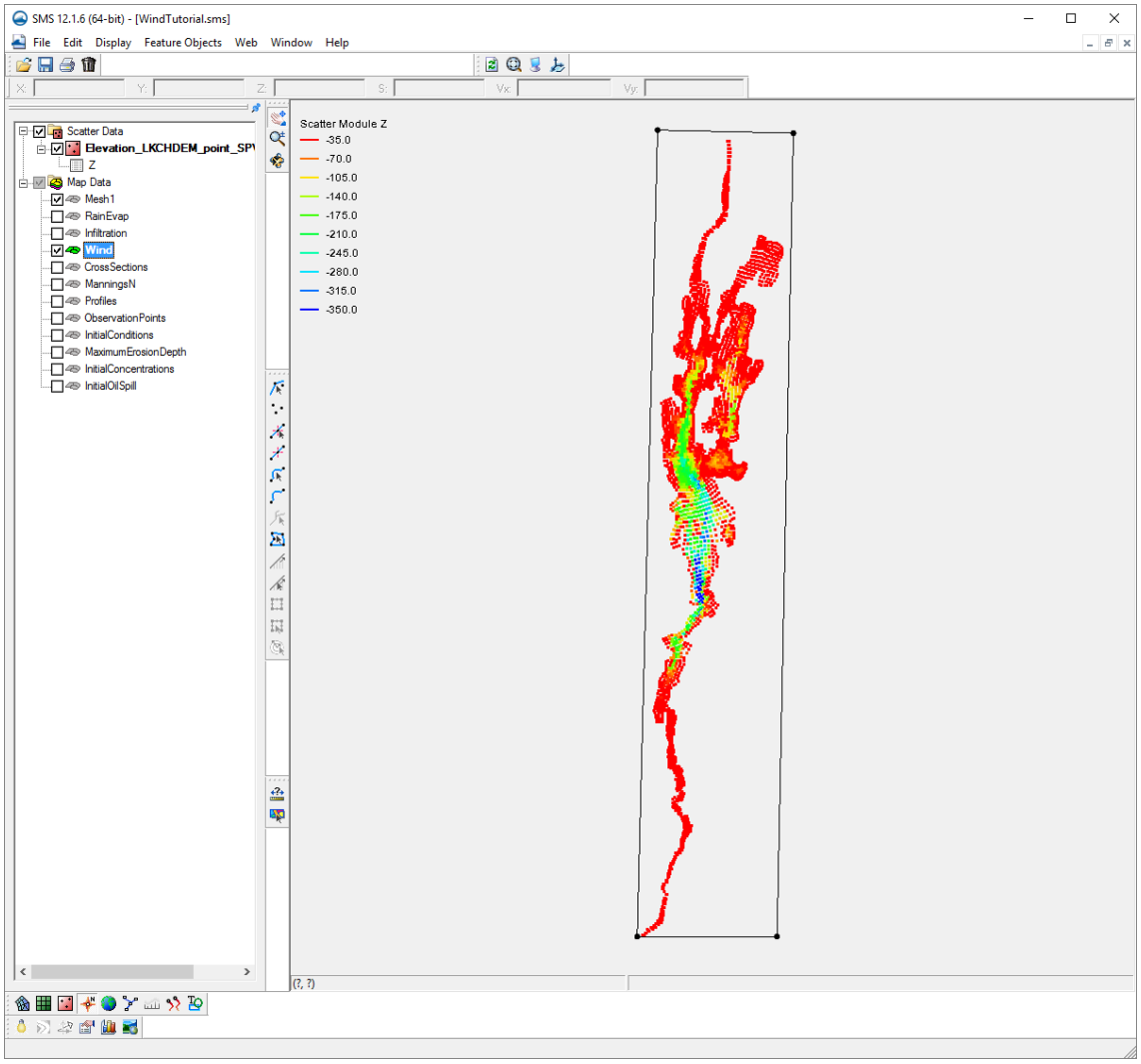


Figure 12.3 – Wind polygon.

In this tutorial we assume that the wind velocity is uniform over the whole mesh area. However, in other cases you may have as many polygons as needed.

3. On the *Feature Objects* menu, click *Build Polygons* to finalize the polygon.

To generate the data structures correctly, the polygon needs to be defined by nodes instead of vertices. To enforce this, click on the *Select Feature Vertex* icon  , on the *Edit* menu, click *Select All*, then right-click on the plot window. On the pop-up menu, click *Convert to Nodes* to convert all vertices to nodes.

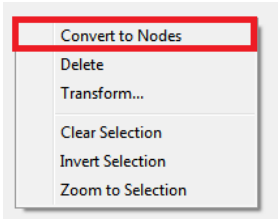


Figure 12.4 – Convert vertices to nodes.

4. Now, click the *Polygon Selection* icon  , click inside the polygon to select it, and right click. On the pop-up menu, click *Attributes*:

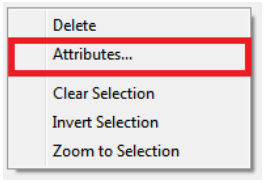


Figure 12.5 – Assign wind velocity data file name to polygon.

5. In the dialog select *Material*, `WindVelocDATA.DAT` from the drop-down material list, and click OK:

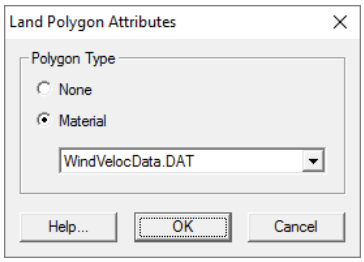


Figure 12.6 – Select wind velocity data file name for polygon.

This completes the creation of wind velocity polygons.

12.5 Generate the Mesh and Copy Wind Velocity Files to the Mesh Folder

To finalize the spatial data preparation, click on the *Mesh* coverage under *Map Data*, and use the *Map* → *2D Mesh* on the Feature Object menu to generate the mesh. Then copy `WindVelocDATA.DAT` to the mesh folder. In this tutorial this folder should be `...\ExampleProjects\WindTutorial\WindTutorial\RiverFlow2D\Mesh1`.

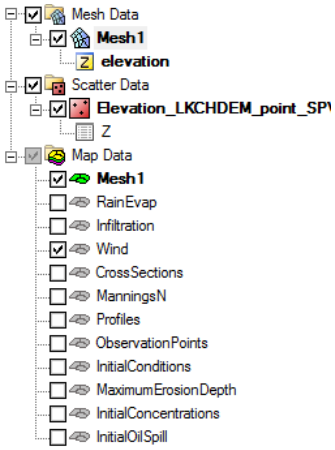


Figure 12.7 – The project folder pointing to the mesh coverage.

12.6 Running the Model

1. Click on the *Mesh Data* Coverage,
2. Click *Run* ***RiverFlow2D*** on the RiverFlow2D menu.

> Please, ignore the *Mesh boundary conditions incomplete warning. It just indicates that you have no open boundaries in this mesh, which is correct in this case since this is a lake.*

3. The Data Input Program will appear.
4. Select Wind on the *Control Data* panel, set the Simulation time to 4 hours, Output interval to 0.1 hours, and Horizontal water surface elevation as shown in Figure 12.8.
5. Save the project.

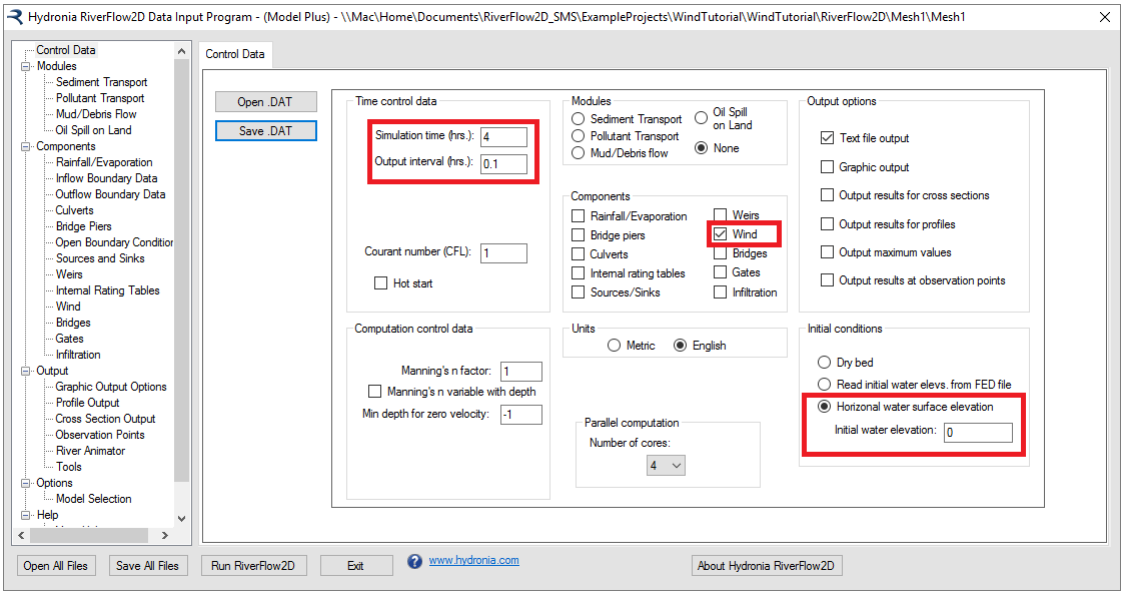


Figure 12.8 – Control Data panel.

6. Finally, click the *Run RiverFlow2D* button. Figure 12.9 shows the dynamic window at the end of the run.

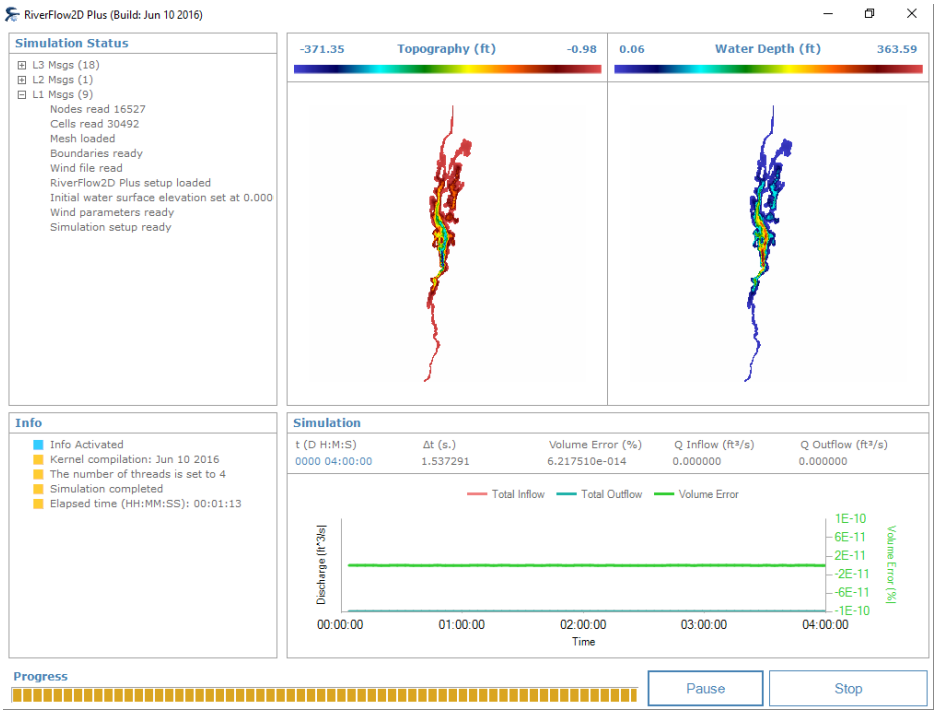


Figure 12.9 – RiverFlow2D window at the end of the Wind Simulation tutorial run.

7. Close the window using the *Close Window* icon on the upper right of the window.

12.7 Visualizing Wind Driven Circulation Results

When the model completes the execution and has generated all of result files, you can use SMS visualization tools to explore results. This is accomplished by opening the `Mesh_RESULTS.HDF5` file that the model creates in the project folder. For instance, you may create velocity field plots as shown in Figure 12.10.

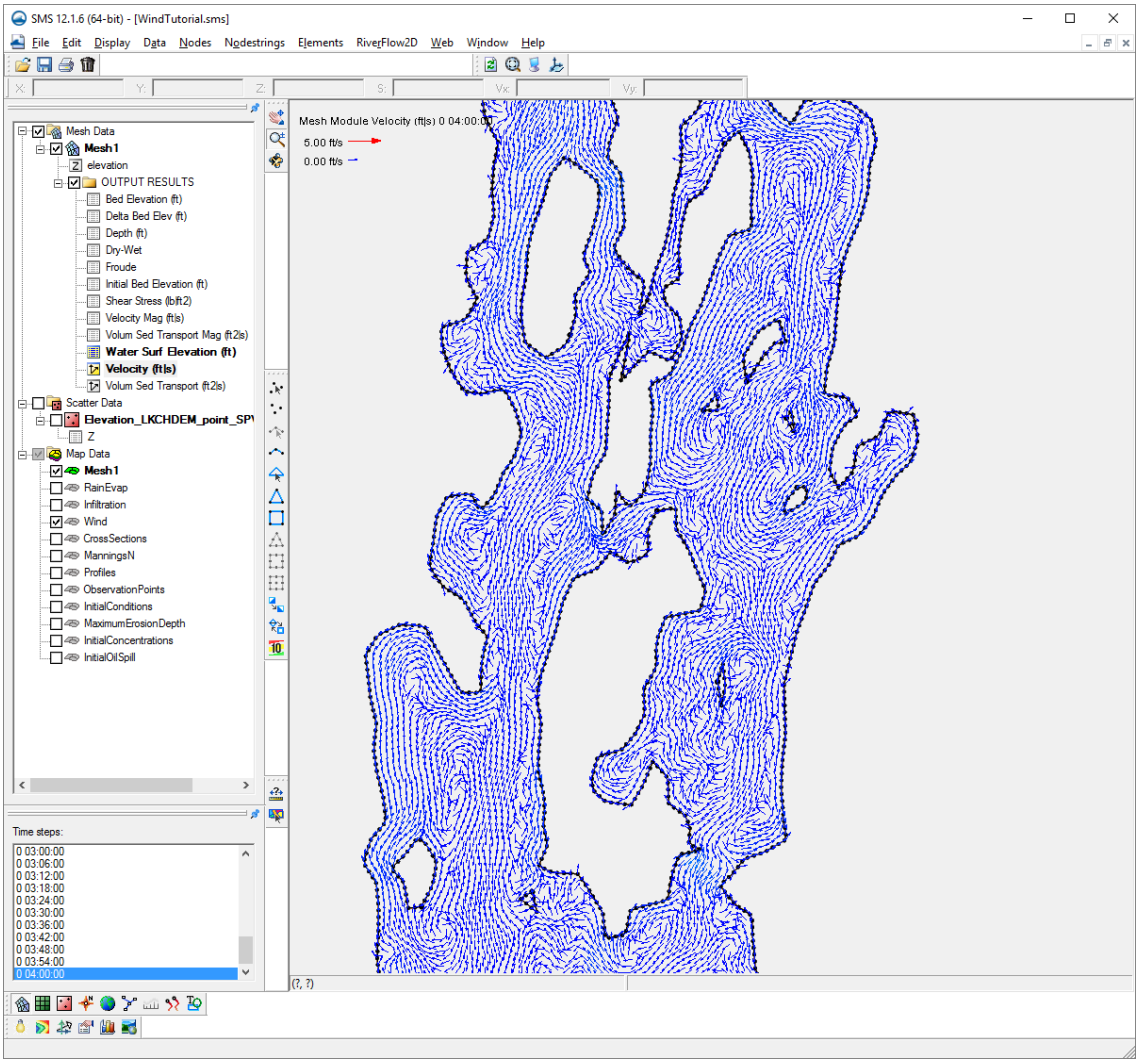


Figure 12.10 – Visualization of wind simulation results.

13 Simulating an Internal Dam Breach

This tutorial will show you how to add an internal dam to you project and simulate a dam breach with the RiverFlow2D *Dam Breach* Component. The procedure involves the following steps:

1. Create the dam breach time series data file.
2. Open an existing RiverFlow2D project.
3. Enter the dam polyline.
4. Enter breach data.
5. Enter the initial water surface elevations.
6. Generate the mesh.
7. Run the model.

The files required to follow this tutorial can be found in the following directory:
`...\Documents\RiverFlow2D_SMS\ExampleProjects\DamBreachTutorial`

13.1 Create a Breach Time Series Data File

Figure 13.1 shows the front view of the schematic dam breach that we want to incorporate to the model. The breach time series file includes the time variation of the breach width b and height H_b for various times.

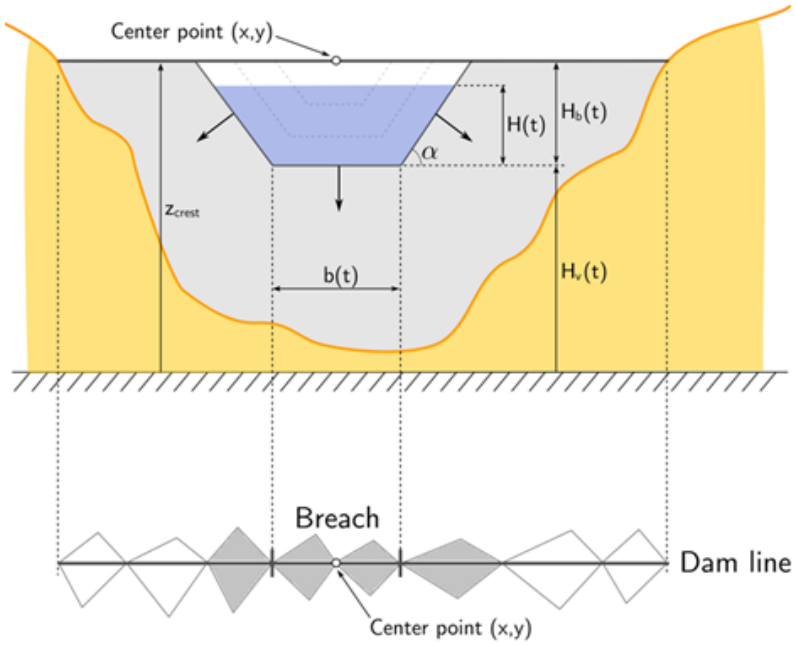


Figure 13.1 – Dam Breach Representation in RiverFlow2D.

This file can have any name. The first line represents the number of times or lines in the file, and then follows as many lines with the time (hrs) b (ft or m) and height H_b (ft or m). For instance the following file has 3 times: 0.0, 0.5 and 24.0 hours. For time 0, the breach width and heght is 0, so the breach has not started to form. Then for time 0.5 hours the breach width is 20m and the breach height is 2m. For intermediate times the model interpolates between the previous and following times.

```
3
 0.0  0.0  0.0
 0.5 20.0  2.0
24.0 20.0 14.0
```

You may use the *Dam Breach* panel in RiverFlow2D Data Input Program to create a dam breach time series file and lets you edit the time, width and height values (see Figure 13.2).

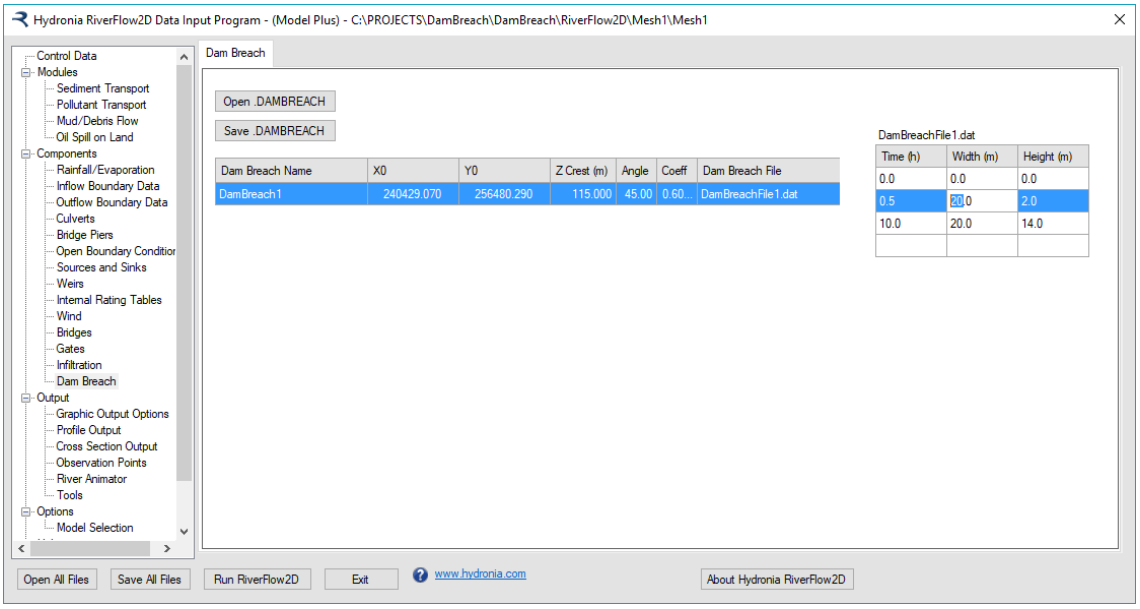


Figure 13.2 – Dam Breach panel in the RiverFlow2D Data Input Program.

13.2 Enter the Dam Polyline in the Default Map Coverage

This step ensures that the mesh will conform to the dam alignment with nodes generated along the dam, cells at each side of the polyline. In this case we will enter the dam is a straight line as follows:

1. Click on the default map coverage.
2. Using the *Create Feature Arc* draw the polyline at the dam location.
3. Select the polyline using the *Select Feature Arc* and redistribute the vertices so that the spacing is 50 m. You should see the dam polyline as shown in Figure 13.3.

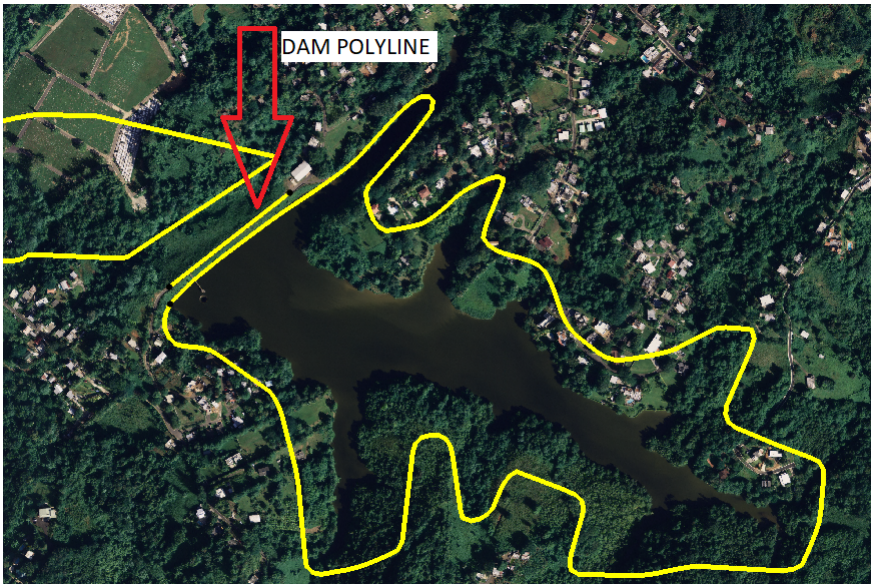


Figure 13.3 – Dam polyline in the main map layer.

4. Now select the polyline again using the *Select Feature Arc*.
5. Right click and select *Attributes...*

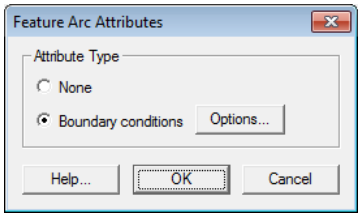


Figure 13.4 – Feature arc attributes dialog.

6. Select Boundary conditions and click Options.
7. Select *Dam Breach* and enter the data as shown in Figure 13.5 and click OK.

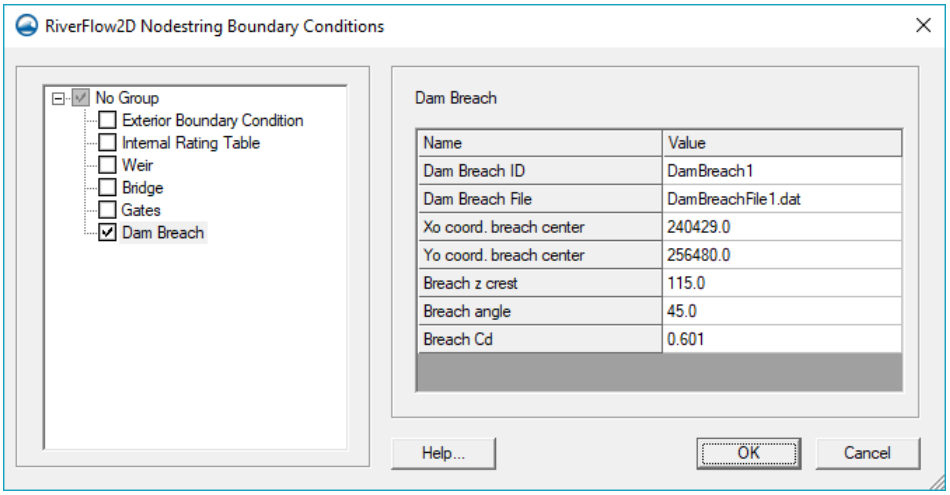


Figure 13.5 – Dam breach boundary conditions dialog.

13.3 Enter the initial water surface elevations

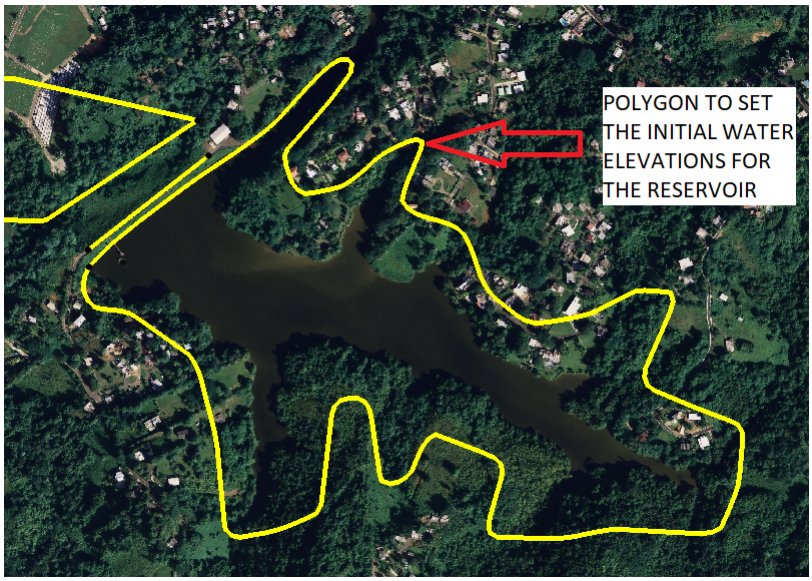


Figure 13.6 – Polygon used to set the initial water elevations on the reservoir.

Use the *Material Data Command* in the *Edit* menu to create new *Material Data* entries with numeric values that will correspond with the initial water elevation that you want to associate with each polygons. To add new values, use the *New* button, and enter Material values. In this case we will enter `115.0`:

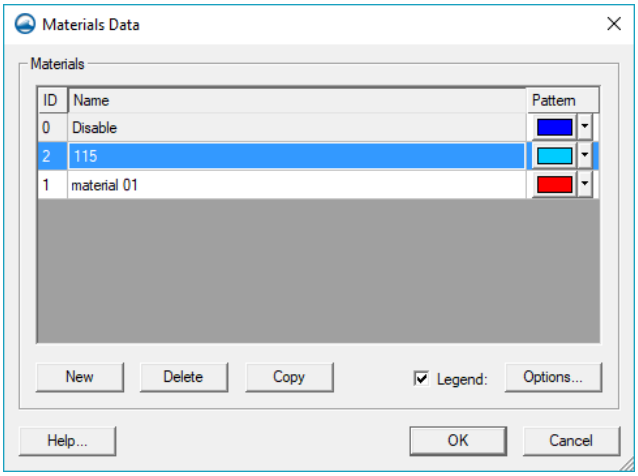


Figure 13.7 – Materials Data Dialog to enter initial water elevation.

We will create an initial water elevation polygon in the *InitialConditions* area coverage.

Note that the *InitialConditions* coverage name should not be changed. If you use a different name for this coverage, the model will not be able recognize the polygons.

To create the polygon, first click on the on the *InitialConditions* coverage to make sure it is the active layer, and using the *Create Feature Arc* tool, draw a polygon as shown:
On the *Feature Objects* menu, click *Build Polygons* to finalize the polygon.

To generate the data structures correctly, the polygon needs to be defined by nodes instead of vertices. To enforce this, click on the *Select Feature Vertex* icon  , on the *Edit* menu, click *Select All*, then right-click on the plot window and on the pop-up menu, click *Convert to Nodes* to convert all vertices to nodes.

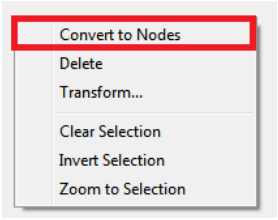


Figure 13.8 – Convert vertices to nodes.

Use the *Polygon Selection* icon  , click inside the polygon to select it, and right click. On the pop-up menu, click *Attributes*:

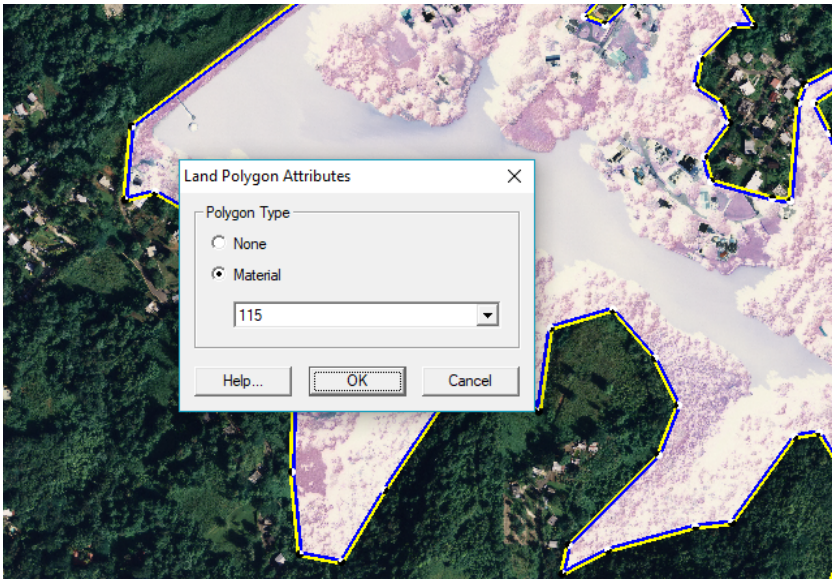


Figure 13.9 – Assign a water elevation value to polygon.

In the dialog select *Material*, `60` from the drop-down material list, and click OK:

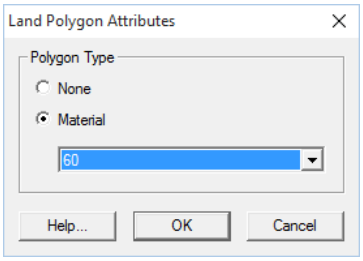


Figure 13.10 – Select value for polygon.

This completes the creation of the water elevation polygons.

13.4 Generate the Mesh

To finalize the spatial data preparation, use the *Map* → *2D Mesh* on the Feature Object menu to generate the mesh. Check that the resulting mesh is perfectly aligned with the dam as shown in Figure 13.11.

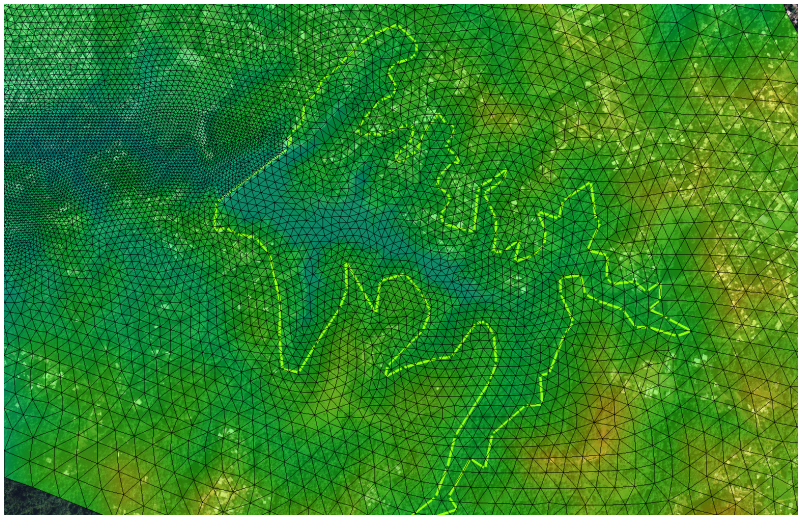


Figure 13.11 – Mesh aligned with dam polyline.

13.5 Running the Model

Now that you have generated the unstructured mesh you need to set up RiverFlow2D run control data and other non-spatial model parameters.

1. Select the *Mesh Data* in the Project explorer to make the *Mesh Module* the active module.
2. On the *RiverFlow2D Menu* menu, click *Global Parameters*

   This will bring up the RiverFlow2D Global Parameters dialog in which you can set the run control parameters for RiverFlow2D.
3. Make sure that the *Dam Breach* component is selected n the *Control Data* panel:

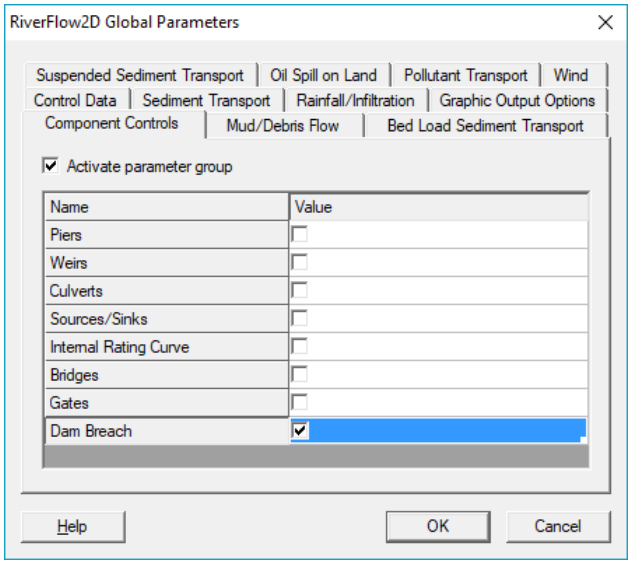


Figure 13.12 – Dam Breach Component.

4. On the *Control Data* tab, enter the options as shown below. Note that the *Initial Conditions* should be set as *Define Initial WSE as Dataset* in order to capture the initial water surface elevation that was specified for the reservoir in the *InitialConditions* coverage

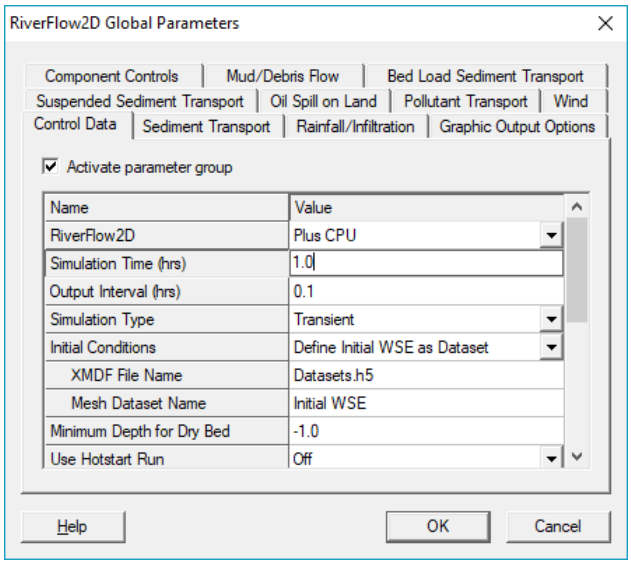


Figure 13.13 – Dam Breach Control Parameters.

Leave all other parameters at their default values.

To run the model, click *Run* ***RiverFlow2D*** button on the lower section of the Data Input Program. A window will appear indicating that the model has started running. The window also reports the simulation time, volume conservation error, total inflow and outflow discharge, and other parameters as the run progresses.

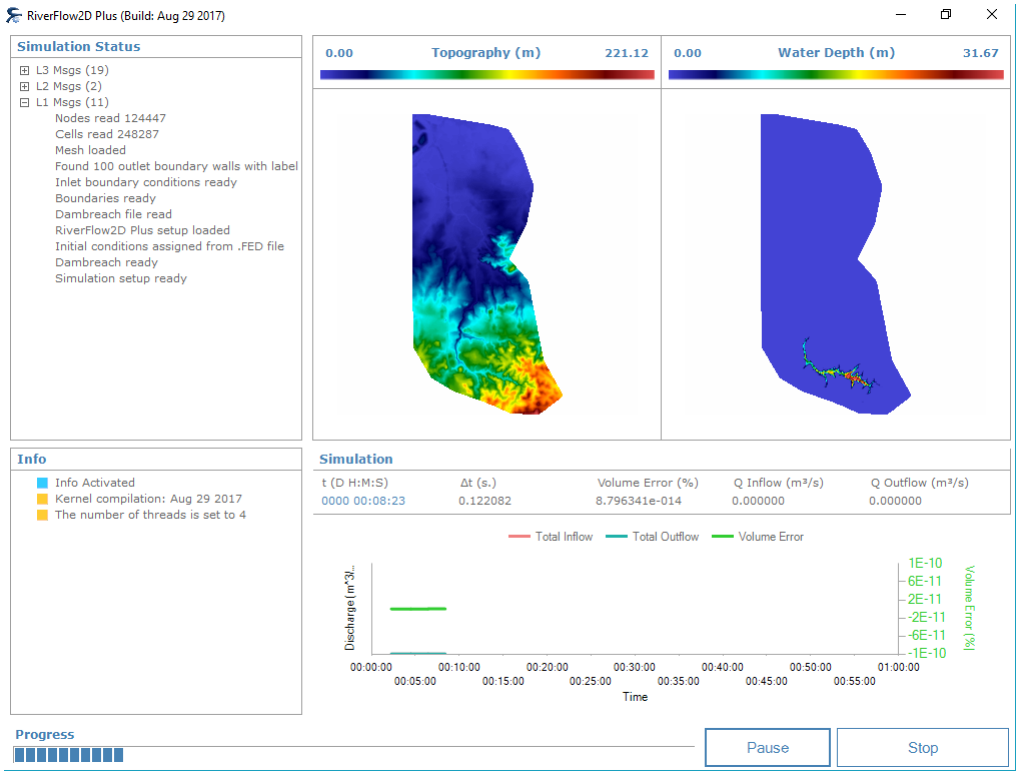


Figure 13.14 – Dam Breach Run Window.

Note that the Simulation Status frame indicates that the Dambreach file has been read and that the Dam Breach component is ready before the simulation started.

14 Post-processing Calculations

14.1 Creating an Observation Coverage

Observation coverages can be created to facilitate the model verification and calibration process, allowing users to verify simulation results with observed data.

To create observation coverage:

1. Right-click the *Map Data* item in the *Project Explorer*.
2. Click *New Coverage* on the pup-up menu.
3. Set the coverage type to *Generic | Observation*.
4. Enter the coverage name.
5. Click OK to exit the dialog.

There are two tools that can be used in observation coverage:

- Observation points are created at locations where calibration data such as the velocity or water surface elevation has been measured in the field. Each observation point is used to compare the measured values with the values computed by the model at the x, y location. Creating an observation point is just like creating a feature point in any other coverage type:
  1. Select the *Create Feature Point* tool from the Toolbar.
  2. Click the location for the feature point.
- Observation arcs, to view the results at a cross section or along the river profile. You can also compute fluxes across the arc that can be compared to measured discharge. Creating an observation arc is just like creating a feature arc in any other coverage type:
  1. Select the *Create Feature Arc* tool from the Toolbar.
  2. Click to create the arc.

3. Double-click to end the arc.

To create observation arc from a shapefile:

1. Bring in a shapefile with the cross-sections.
2. Convert the shapefile to feature objects to get arcs.
3. Double click the arc.
4. Choose the module: scalar as water depth or vector as velocity.
5. Discharge is reported for each arc in the observation arcs box.

14.2 Cross Sections

Cross sections or profiles can be created using observation arcs in SMS. Arc attributes in SMS can be set up once the model has completed a run and the results are read back into SMS. To create a cross section of the river:

1. Right click on the *Map Data* object in the *Project Explorer* and create a new coverage. Make the type an Observation coverage by right clicking and selecting *Type* | *Generic* | *Observation* and make its name 'Observation Plots'. Click on this new coverage to make the Map module active. (SMS creates a separate coverage to keep the observation arcs separate from the exiting observation points. When an observation arc is being created, observation points may be clicked joining them to the arc. Observation points and arcs can exist on the same coverage.)
2. Select the *Create Feature Arcs* tool from the *Toolbox*.
3. With the cross sections created, open the *Observation Coverage* dialog and assign a unique color and an appropriate name to each arc.

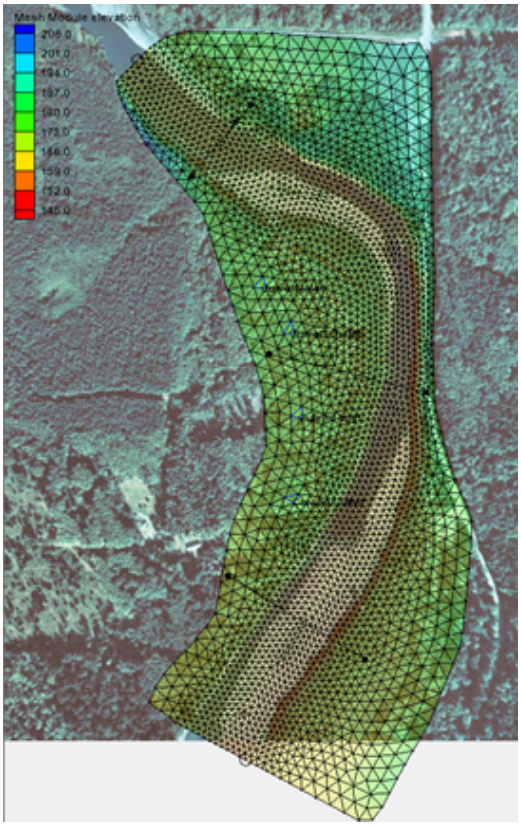


Figure 14.1 – Cross sections arcs on the Observation coverage.

With the arcs created, the plots can now be generated as follows:

1. On the *Display* menu, click *Plot Wizard...*
2. Choose *Observation Profile* as the *Plot Type* and click Next.
3. Turn on the *Specified* option, and check only the Bed elevation data set in the Generic Solution and the water surface elevation data set in the .hdf5 solution.
4. Turn off the three cross-section arcs in the Arcs spreadsheet by turning off their corresponding Show toggles.
5. Click Finish to close the *Plot Wizard* and generate the plot.

The profile plot of the geometry of the stream will of appear as shown in Figure 14.2.

To view the velocity distribution across the three cross sections:

1. Create a new *Observation Profile* plot.
2. Turn on the *velocity mag* data set in the solution.
3. *Show* only the three cross section arcs.

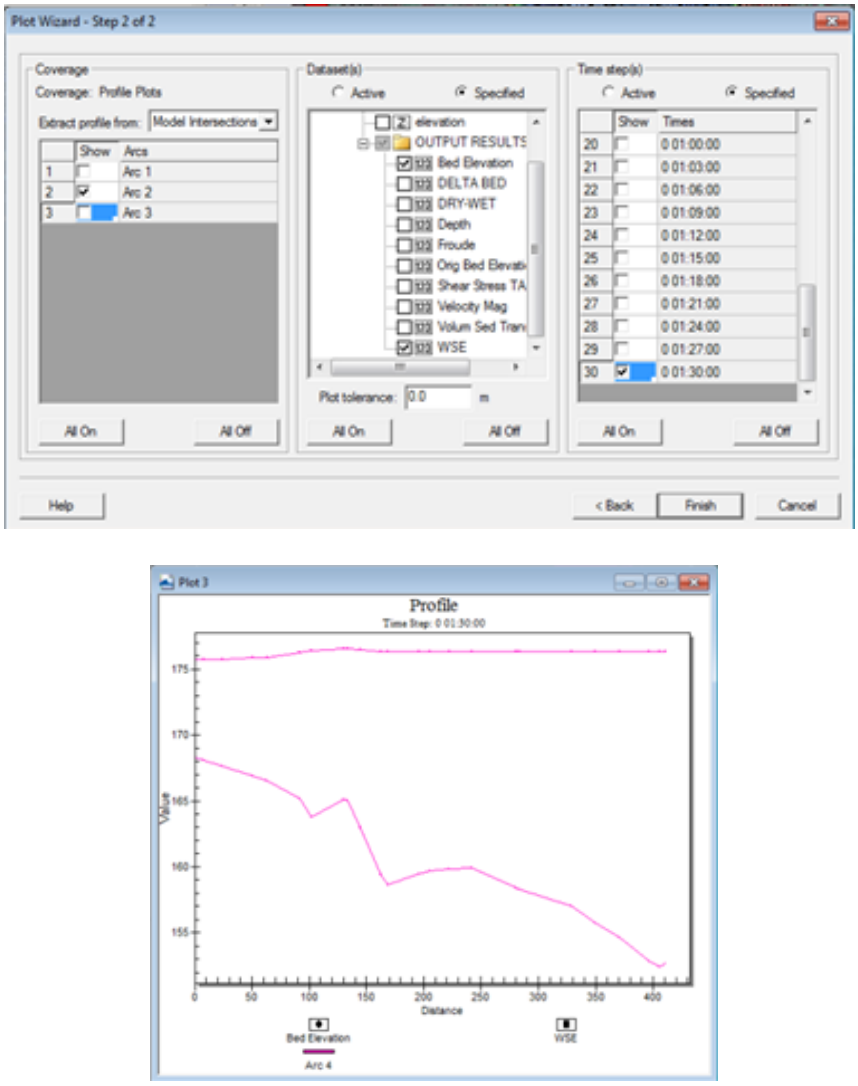


Figure 14.2 – Profile plot of the geometry of the stream.

14.3 Profiles

Profiles can be created using observation arcs in SMS. Profiles attributes in SMS can be set up once the model has been run and the results are read. To create a profile of the river:

1. Right click on the *Map Data* object in the *Project Explorer* and create a new coverage.

2. Make the type Observation coverage and make its name Profiles'.

3. Click on this new coverage to make the Map module active. User can create a separate coverage to keep the cross section arcs separate from the exiting profiles. Profiles and cross section arcs can exist on the same coverage.

4. Choose the *Create Feature Arc* tool from the Toolbox.

5. Create an arc down the main channel. Remember to double-click the last point to end the arc.

When the plots are drawn, they will use the name and color associated with the observation arc. To change the name and color of the arc:

1. Choose the Select Feature Arcs tool from the Toolbox.
2. Double-click on the profile arc.
3. In the *Observation Coverage* dialog, change the Name of the arc to 'river profile' in the *Observation Arcs*.
4. Click OK to close the *Observation Coverage* dialog.

With the arcs created, the plots can now be generated. To do this:

1. On the *Display* menu, click *Plot Wizard...*
2. Choose *Observation Profile* as the *Plot Type* and click Next.
3. Turn on the *Specified* option, and check only the bed elevation data set in the Generic Solution and the water surface elevation data set in the .hdf5 solution.
4. Turn off the additional arcs in the Arcs spreadsheet by turning off their corresponding Show toggles and turn on the arc corresponding with the profile.
5. Click Finish to close the *Plot Wizard* and generate the plot.

15 Interpolation Methods

The interpolation methods in SMS can be call on as:

1. Explicit procedure: user selects to interpolate a scatter dataset to a mesh or to a map, by selecting the Scatter Menu in the Scatter Module. The Scatter Set should already exist in the project as well as the mesh or map to which the interpolation will be linked. The interpolation control dialog can be accessed from the Scatter Menu by clicking on one of the three options:

   - Interpolate to Mesh
   - Interpolate to Scatter
   - Interpolate to Map

2. Implicit procedure: is part of the automatic mesh generation and it was explained already in Section 5 of this Manual. The interpolation control dialog can be accessed from the Conceptual Model Coverage by selecting the polygons using the Select Feature Polygon  , right clicking on the polygon and click on Attributes, the Mesh Polygon Properties will open, the interpolation dialog will open by clicking on Scatter

   The options to select in the interpolation option dialogs can be described as:

   - **New dataset name:** name of the new dataset (or function) created by interpolation.
   - **Map Elevation:** only for interpolation to mesh nodes, the new dataset is mapped to be the elevation function.
   - **Interpolation Method:** the supported interpolation methods in SMS include:
     (a) Linear
     (b) Inverse Distance Weighted (IDW)
     (c) Natural Neighbor (NN)
   - **Default Extrapolation Value:** for linear and NN interpolation methods, if the scatter set does not bound the data being interpolated to, the extrapolation value is used.
   - **Existing Dataset Value:** corresponding value from a specified existing dataset can be used for locations outside of the bounds of the scatter set. The dataset must be from the same object being interpolated to and must be of the same type (i.e. scalar, vector).

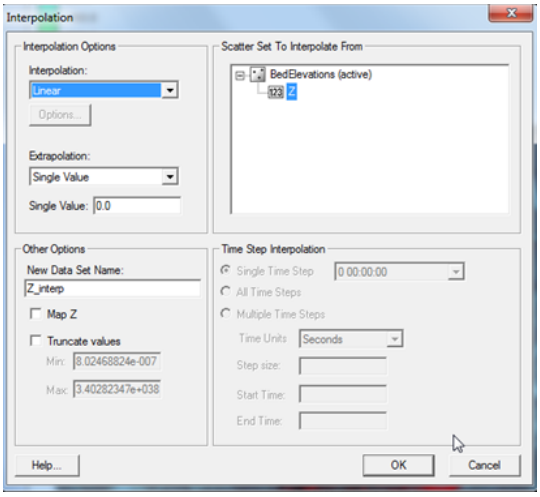


Figure 15.1 – Interpolation Control dialog.

- **Truncate Values:** when interpolating a set of values, it is sometimes useful to limit the interpolated values to a specific range. Depending on the application different meaningless values can be truncated. Sometimes, an interpolation scheme will produce negative values even if all of the scatter points have positive data values. This occurs in areas where the trend in the data is toward a zero value. The interpolation may extend the trend beyond a zero value into the negative range. In such cases it is useful to limit the minimum interpolated value to zero. Interpolated values can be limited to a given range by entering a minimum and maximum interpolation value.
- **Scatter Set/Function:** user selects the desired scatter point set and the function to interpolate from.
- **Time Steps:** if the function is transient the user selects the option to interpolate a single time step or multiple time steps.
- **Time Step Interpolation:** if interpolating multiple time steps, the user can select the beginning time, the step size, and the number of time steps to interpolate. The user can also choose to interpolate between time steps or to match time steps that fall within the specified time range.

16 Using Huge Elevation Data Sets

The way to import and filter huge elevation point datasets is as follows:

1. Use the import wizard tools to only bring in a portion of the data generally divided up by area (bounding box). Second step of the import wizard choose the filter options button.
2. Filter the scatter set using the filter tool in SMS. Scatter module choose *Data* | *Filter*.
3. Save the filtered data to a SMS file.
4. Delete the scatter set.
5. Repeat steps 1-4 for different areas until all of the points you are interested in have been filtered.
6. Load each of the files and merge them into a single scatter set. The best way to do this might be to start with the first scatterset, load another file, merge it into the merged scatter set, delete it, and repeat. Doing this SMS will only have the merged scatterset and one other scatterset at the same time.
7. If the total number of points is still too large, you may have to filter the individual scattersets further.

ASCII Elevation data files with more than 750 million of points will need to be filtered using the SMS Scatter module. The points can be filtered within the import wizard in SMS. Processing and filtering that many points will require a lot of memory and CPU time.

Another option to import large elevation datasets available from SMS v11.0 is to use the new Raster Module that is designed for handling elevation datasets with many more points than what are typically found. Each point in the Raster module requires much less memory than the Scatter module since the data are on a regular grid and SMS doesn't have to store triangulation data. Data in the Raster module can also be interpolated to a mesh.

17 Converting .DXF Files to Scatter Data

Drawing Exchange Format (DXF) files are AutoCAD files that contain vector drawing data. They can be imported for background display or for conversion to feature objects as a scatter data set. The following steps summarize the procedure that has to be following to convert a DXF to a scatter data set:

1. Select *File* | *Delete All* to clear out the data in SMS.
2. *Select File* | *Open*.
3. Open the file .dxf. Project Explorer will display the CAD layer with a set of contours.
4. Right-click on CAD layer in the Project Explorer and select Convert → CAD → Map from the drop-down menu. A new coverage named CAD will be created. Select this coverage to make the Map Module active.
5. Right-click on the CAD coverage and select Convert → Map → 2D Scatter.
6. Click OK in Map → Scatter dialog. This process can take a few minutes since the spacing of the points along the contours is high resolution.
7. Turn off the CAD map coverage and zoom in to the scatter set.

18 Using HEC-RAS Data Extraction Tool

This tutorial explains how to use the HEC-RAS Data Extraction Tool to read elevation data from HEC-RAS cross sections and use the elevations in RiverFlow2D.

The steps used in this tutorial extract HEC-RAS elevation data and use it in RiverFlow2D are as follows:

1. Use HEC-RAS Data Extraction Tool to read the geometry file and create the X Y Z and polygon files.
2. Open SMS and import new data for the elevation points.
3. Import the elevation points.
4. Link the bed elevation parameter to the HEC-RAS data layer.
5. Enter the modeling area.
6. Generate the mesh.
7. Check interpolated elevations.

The files required to follow this tutorial can be found in the following directory:
`...\Documents\RiverFlow2D_SMS\ExampleProjects\HEC-RASDataExtractionTool`

18.1 Read HEC-RAS Geometry File and Create the X Y Z and Polygon Files

The first step is to open the RiverFlow2D Data Input Program (DIP) and click the Tools panel to access the HEC-RAS Data Extraction Tool:
Click on Extract Data from HEC-RAS and open the geometry file WAILUPE.g02 in the folder:

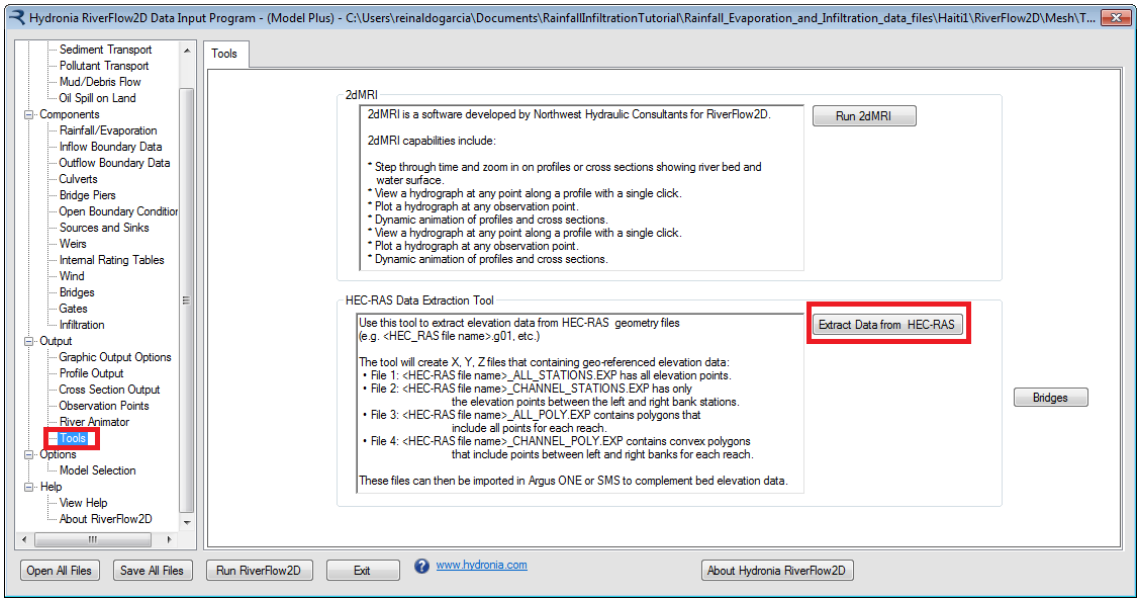


Figure 18.1 – Extract data from HEC-RAS command on Tool panel.

...\Documents\RiverFlow2D_SMS\ExampleProjects\HEC-RAS Data Extraction Tool

The program reports the number of cross sections, cut-lines and banks found:

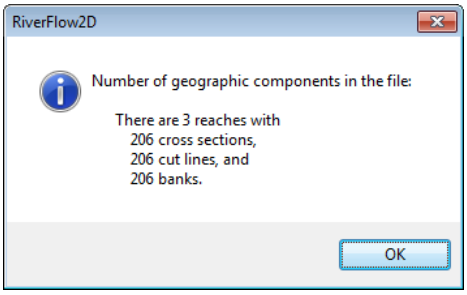


Figure 18.2 – Results of HEC-RAS data extraction.

Click OK, and select the directory where you want the output files. In this case, we will put these files in the same directory of the HEC-RAS geometry file:

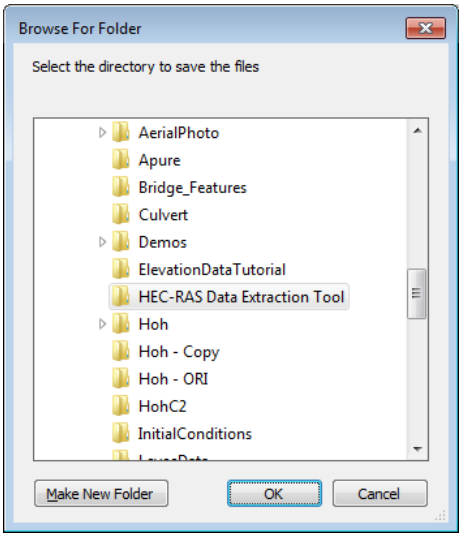


Figure 18.3 – Folder for HEC-RAS extracted files.

Click OK and the following dialog reports that the output files were created:

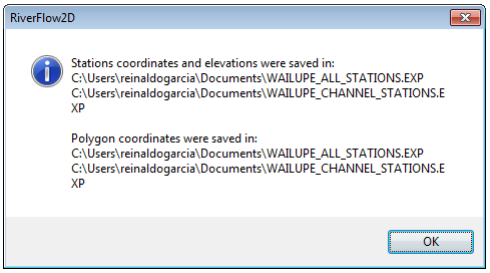


Figure 18.4 – Confirmation of HEC-RAS extracted files.

The files indicated in the following table are generated:

Table 18.1 – Files generated by the HEC-RAS Extraction tool.

`WAILUPE_ALL_STATIONS.EXP`	X Y Z Elevation files containing all points
`WAILUPE_ALL_POLY.EXP`	Polygons that include all points
`WAILUPE_CHANNEL_STATIONS.EXP`	X Y Z Elevation files containing points in the main channel only
`WAILUPE_CHANNEL_POLY.EXP`	Polygons that include points in channel

The generated X,Y,Z can be imported in SMS to complement other elevation data.

19 Investigating the Mesh Quality

Mesh quality can be checked on an existing mesh in SMS:

- On the *Display* menu, click *Display Options*.
- Select the *2D Mesh* tab.
- Turn off the Contours and turn on the Mesh quality.
- Click the OK button.

The display will refresh without contours and depicting problematic areas. A legend shows the color corresponding with each quality category.

- Turn off the mesh quality button on *Display* | *Display Options*.

The quality checks that are displayed on the mesh are shown in figure 19.2.

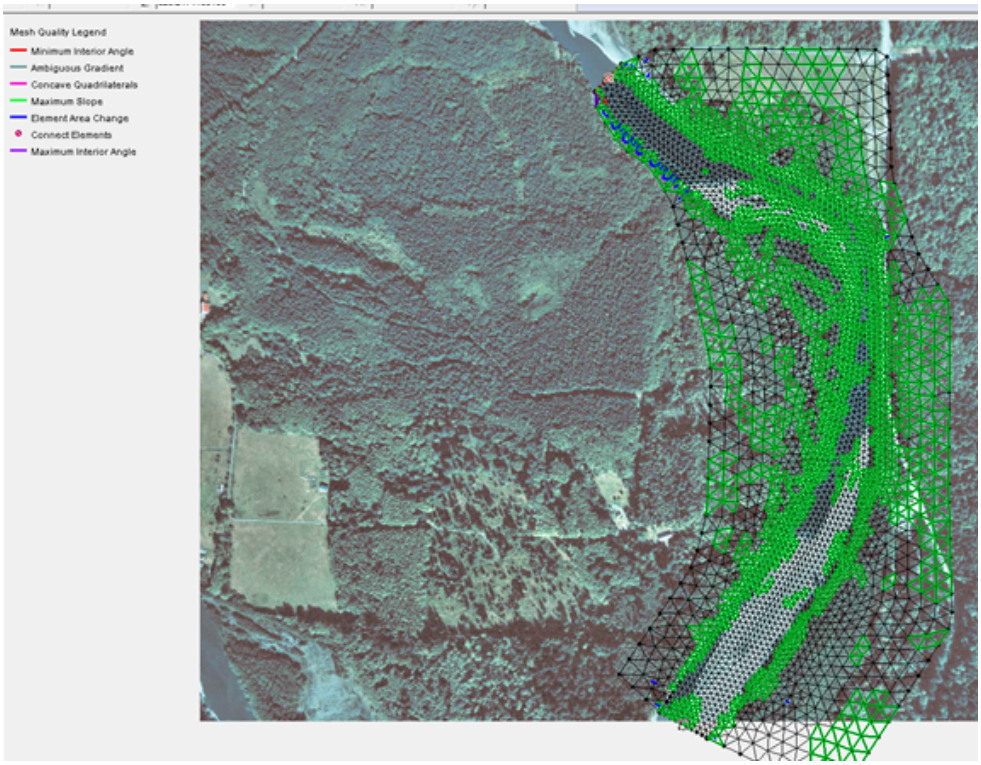


Figure 19.1 – Mesh Quality for Hoh River mesh.

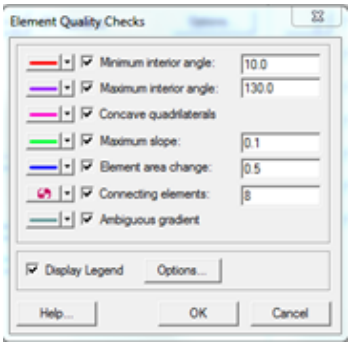


Figure 19.2 – Quality checks.

20 Refining Elements

Sometimes it is needed to increase the mesh resolution on some areas of the mesh. The gradient of the physical quantities on the mesh might be large or there are sensitive areas that might need more detail. It is important to understand that refining the mesh too much, will generate a mesh with more nodes and elements, and that will lead to increased computational time.

20.1 Inserting Breaklines

A nodestring can be used as a breakline to cut elements and refine the mesh. In order to cut the elements, you must create two nodes, and then define a nodestring. Follow the instructions below:

- Zoom into the area to refine.
- Choose the *Create Mesh Node* tool from the *Toolbox*.

After creating the first node, assign the bed elevation value in the edit window to be 171.0. This will assign the elevation of the new nodes to be similar to the existing nodes. A nodestring can now be created from one of these new nodes to the other. This nodestring will be used as a breakline.

- Choose the *Create Nodestring* tool from the *Toolbox*.
- Create the nodestring using the new two nodes.

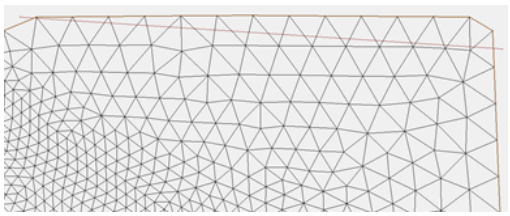


Figure 20.1 – Creating the nodestring to split elements.

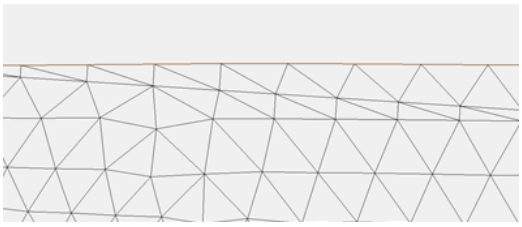


Figure 20.2 – Split elements.

- Select *Nodestring* | *Force Breaklines*. Elements will be split.

After the nodestring is used as a breakline, it is no longer needed and it should be deleted.

- Select *Edit* | *Delete* or click the Delete macro.

20.2 Arc Inserted on the Conceptual Model Coverage

A Feature Arc can be used to force element edges to appear along the line and refine the mesh. In order to force nodes to be along a line, the arc must be created on the conceptual model. Follow the instructions to do this:

- Zoom into the area to refine.

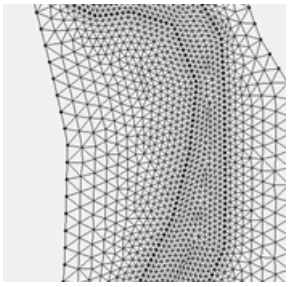


Figure 20.3 – Coarse Elements near the boundary.

- Choose the Create Feature Arc tool from the Toolbox.
- After creating the arc, select Feature Arc, right click.
- Then select Redistribute Vertices, and enter Specified Spacing and Spacing equal to 20 ft.
- There are other options as Number of Segments and Min and Max Spacing that can be used to specify the number of elements or the spacing that will be created.

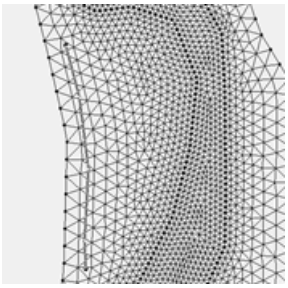


Figure 20.4 – Feature Arc created near to the boundary to refine the mesh.

Then activate the conceptual model and create a new 2D mesh. The areas around the Feature Arc will be refined using the defined spacing:

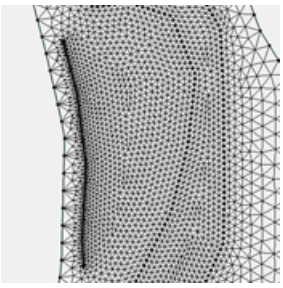


Figure 20.5 – Refine mesh.

The nodes will be forced to pass through the defined feature arc.

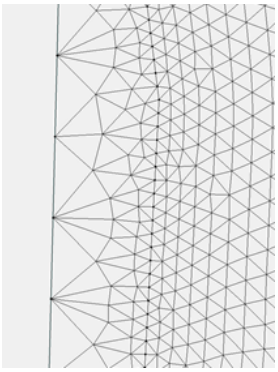


Figure 20.6 – Nodes passing through the Feature Arc.

21 Creating New Data Sets Using the Data Calculator

A new data sets can be computed by performing operations on scalar values in existing data sets. Even though RiverFlow2D generates the Froude number for each report time, in this example we will illustrate the Data Calculator Tool creating a new data set for the Froude Number (F) using the equation:

$$F = \frac{V}{\sqrt{gh}} \tag{21.1}$$

Where

V=velocity magnitude.

g=gravitational acceleration.

h=depth.

Use the following procedure to create the new data set:

1. Select *Data* | *Data Calculator*.
2. Highlight the Velocity Mag data set.
3. Under the Time Steps section, turn on the Use all time steps option and add to Expression button.
4. Click the divide button. Click the sqrt(x) operation. Enter 32.2 for the constant g.
5. Click the multiply button, then highlight the water depth data set and click the Add to Expression button.

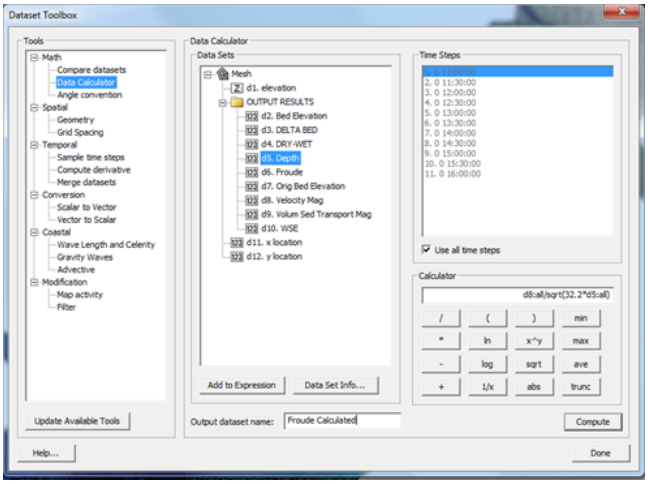


Figure 21.1 – Expression to calculate Froude number.

6. Enter the name of the field in the Output dataset, enter Computation of Froude and then click Compute button. SMS will take a few moments to create the data set.

7. Click the Done button to exit the Data Calculator dialog.

The new data set is automatically placed to a `.sol` file by default, to be edited with any of the other tools in SMS or saved in a generic data set.

22
Exporting Shapefiles

To create a shapefile, just use the Save as Shapefile... Command from the File Menu.

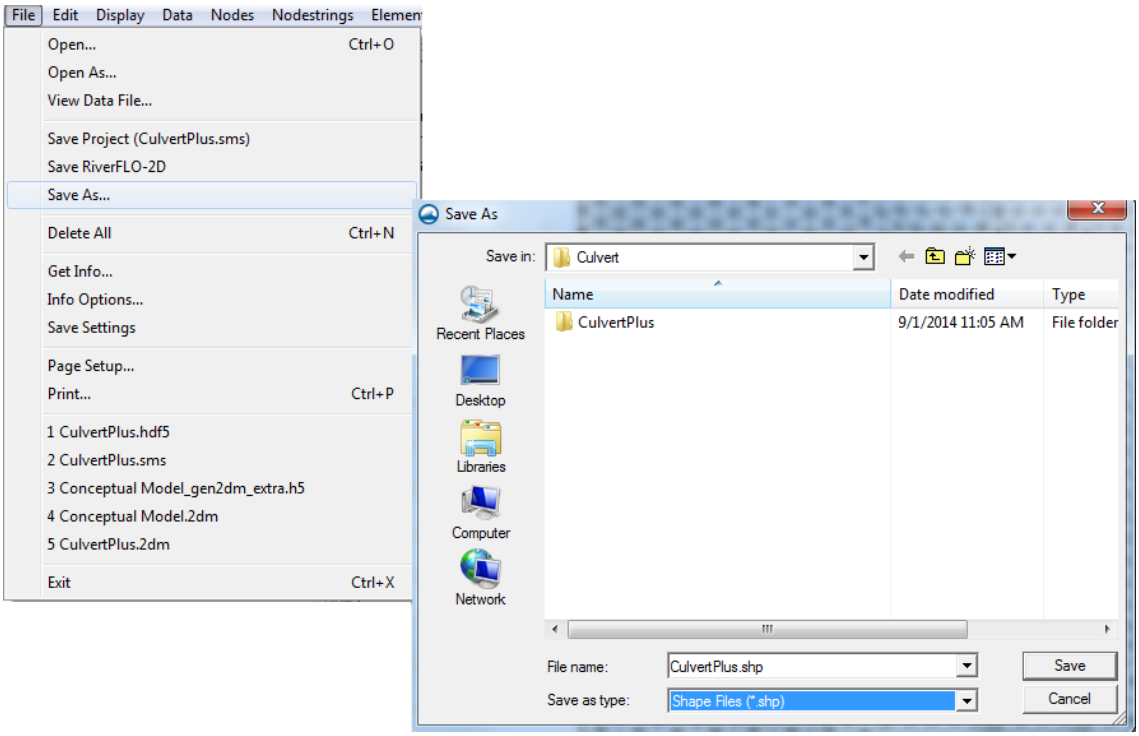


Figure 22.1 – Export shapefile command.

This will create a shapefile on the project directory.