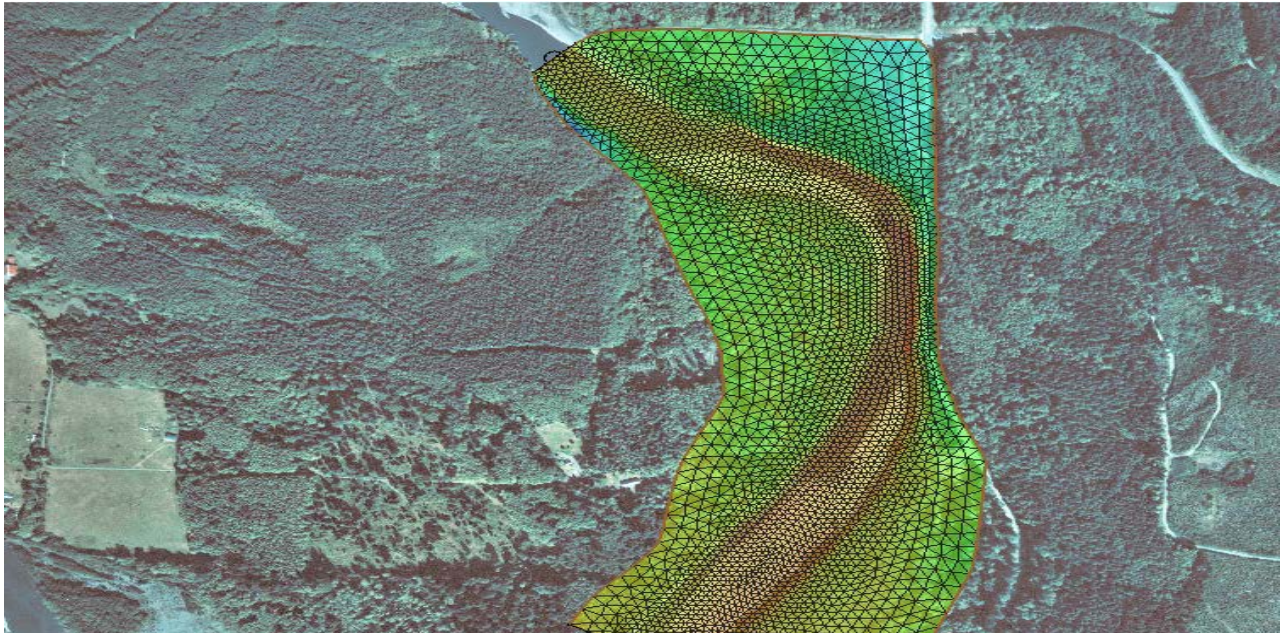


## SMS 11.1 Tutorial

### **RiverFLO-2D Analysis**



#### Objectives

This lesson will teach you how to prepare an unstructured mesh, run the RiverFLO-2D numerical engine and view the results all within SMS. You will start by reading in the RiverFLO-2D Generic Model Interface template and then work with the different initial data files in SMS to create and run your RiverFLO-2D simulation.

The data used for this tutorial is taken from a section of the Hoh River in the state of Washington, USA. This data is from a project where the objective was to analyze the effect of engineered log jam (ELJ) designs on this bend in the river.

#### Prerequisites

- Overview Tutorial

#### Requirements

- RiverFLO-2D
- Map Module
- Mesh Module
- Scatter Module

#### Time

- 45-60 minutes

**AQUAVEO™**



# 1 Getting Started

---

## 1.1 Open Template

For this tutorial, first you must read in the RiverFLO-2D template (.2dm). This template has been developed by Aquaveo and Hydronia in order to integrate the RiverFLO-2D numerical engine into the SMS graphical user interface for pre/post processing. To read in the template file:

1. Run the SMS Program.
2. Select File | Open.
3. Select the file *RiverFLO-2D\_Template.2dm* in the folder for this tutorial and click the *Open* button.

## 1.1 Define Projection and Units

In SMS you can define a projection for your simulation. While RiverFLO-2D can be run using U.S. Customary or Metric Units, setting a projection also allows you to take advantage of all the geo-referencing tools in SMS for pre/post processing. It is a reminder as well to use consistent units throughout your project.

To set the projection:

1. Choose Display | Projection....
2. Set the *Horizontal* component to use a *Global projection*. This will bring up the *Select Projection* dialog. Select *UTM* as the projection and *NAD83* as the datum. Also set the *Planar Units* to *Feet (U.S. Survey)* and the *Zone* to 10. Click *OK* to complete the projection selection.
3. Ensure that the *Vertical* units are set to *U. S. Survey Feet*.
4. Click the *OK* button to exit the dialog.
5. The project has now been set in the UTM projection and the Units will be in U.S. Survey feet.
6. Save your Project by selecting *File / Save As...*
7. Make sure the *Save as type* is Project Files (\*.sms) and enter the name *HohRiver.sms*.
8. Click the *Save* button to save the simulation.

Note: With the RiverFLO-2D the Horizontal and Vertical Units need to be the same. Importing data with different units is possible but the data must be converted to the desired units before running RiverFLO-2D.

## 1.2 Read in Initial Data

The next step is to read in any initial data that you have gathered. For the Hoh River you have a background image of the site and an elevation survey.

Read in Images:

1. Select File | Open.
2. Select the file *HohRiver.jpg* in the data files folder for this tutorial and click the *Open* button.
3. Click *No* 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 the detailed resolution of the image.

Read in Topographic Data:

1. Select File | 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* button 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 open the file. (This wizard allows you to open data that may not have data in 3 columns of x, y, and z. Data in any number of columns in any order can be opened through the wizard).

### 1.3 Display Options

You will want to adjust our display settings to see the elevations contoured.

1. Select Display | Display Options.
2. Toggle off *Points* and toggle on *Contours* and *Boundary* in the *Scatter* display option 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 should appear similar to below in the display window.

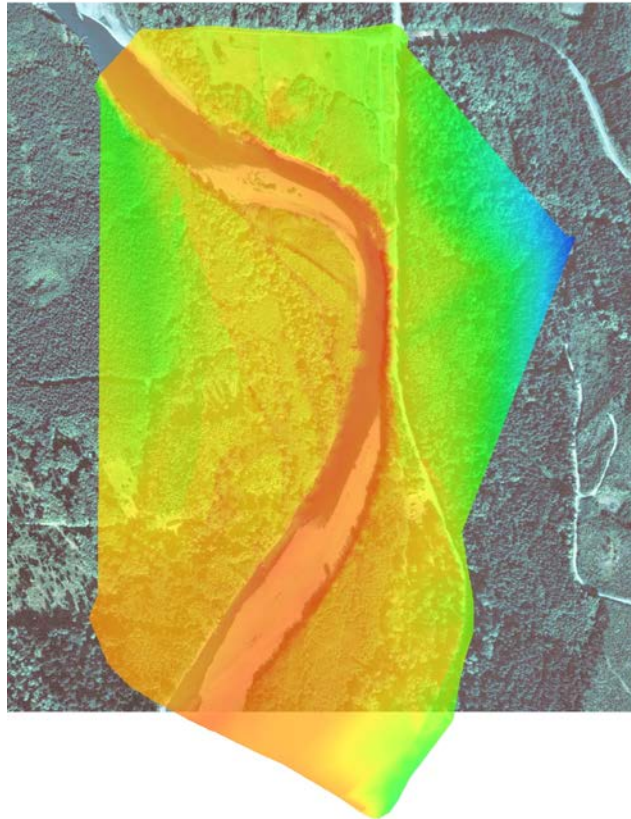


Figure : Elevation Data and Background Image



## 2 Define Model Domain


---

### 2.1 Define Coverage Type

A coverage in the Map Module is where different model features are described using GIS feature objects like points, arcs and polygons. The shape of the modeling domain will be defined in a Generic 2D Mesh coverage.

1. Right click on default coverage and select *Type | Models | Generic 2D Mesh*.
2. Right click on *default coverage* and select *rename*.
3. Change the name of coverage to “*Hoh River*”.

### 2.2 Create Polygons

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 .

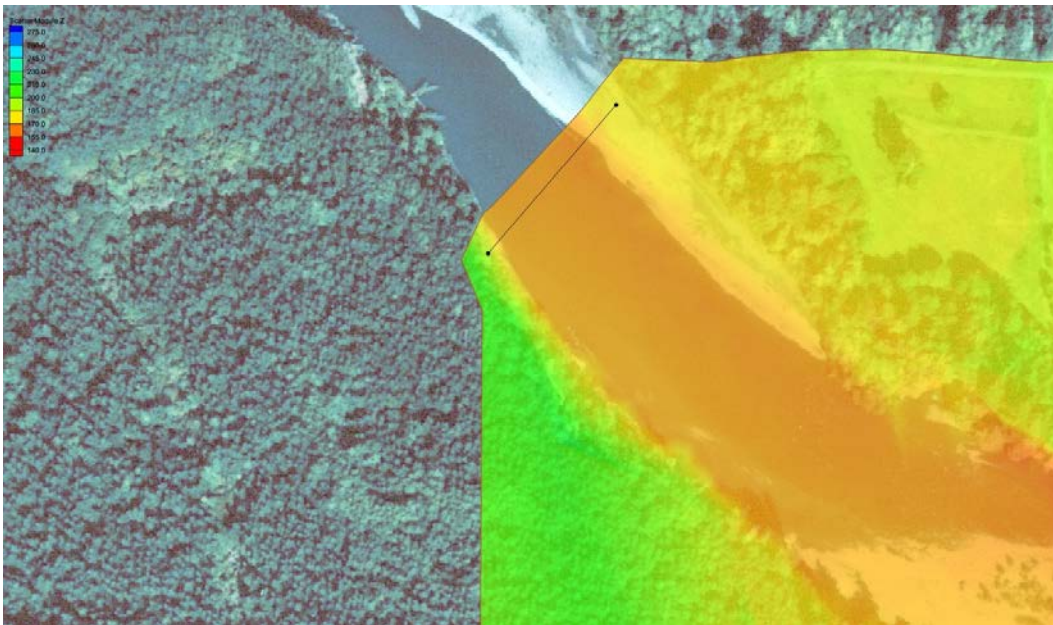
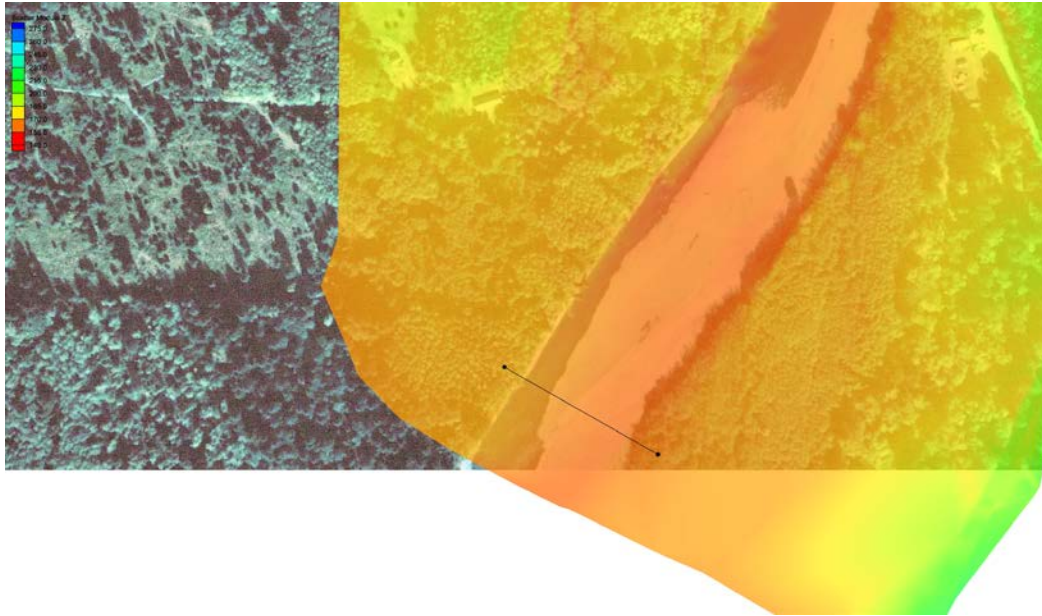


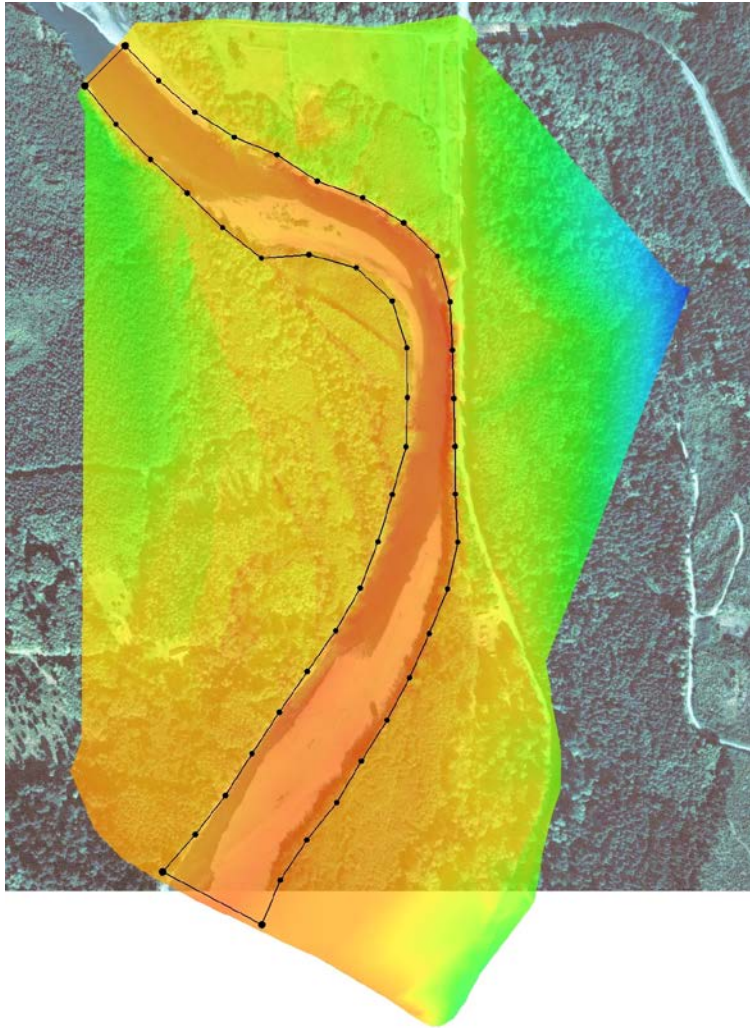
Figure : Upstream Boundary

3. Do the same for the southernmost part of the channel.



*Figure : Downstream Boundary*

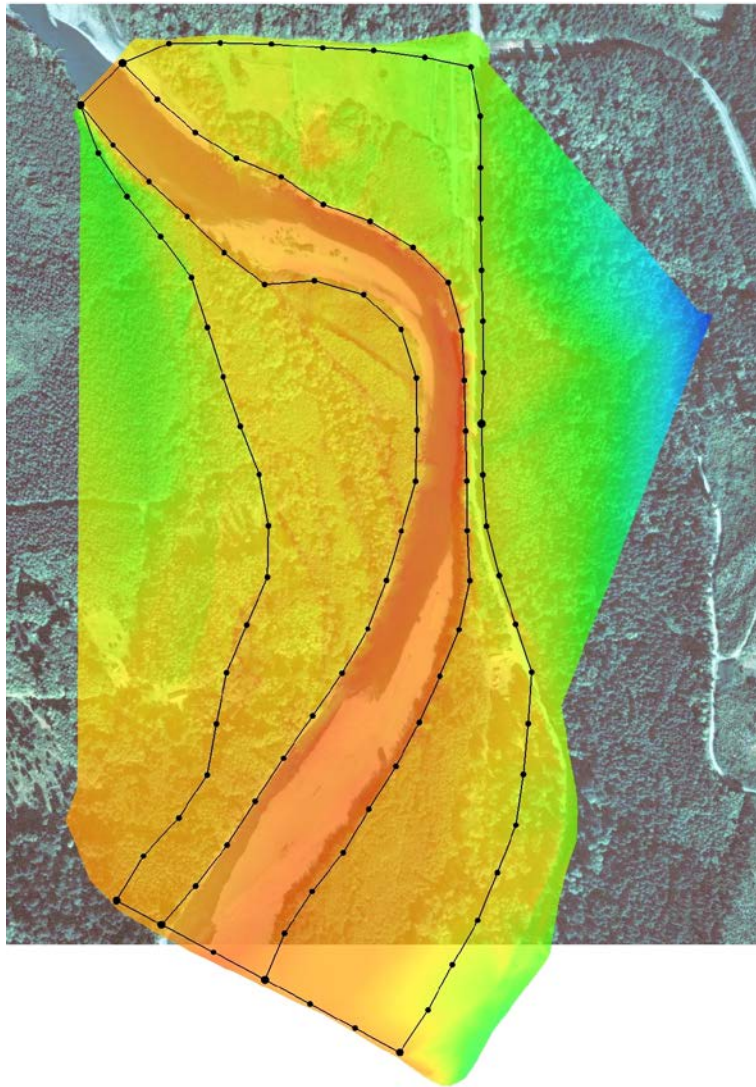
4. Now create feature arcs connecting the two east node and two west nodes following the banks of the channel displayed by the contours



*Figure : Channel Defined*

5. Now create two more arcs, one east and another west of the channel, to define the far eastern and western extents of our domain. These arcs will begin and end on the channel cross section arc we previously created.





*Figure : Domain Extents Defined*

### **2.3 Define Arc Spacing**

The different arcs in our Mesh Boundary coverage can be used to vary the mesh element size. There are a few different tools that you can use in SMS to do this but in this tutorial the spacing between vertices on the arcs will be used to vary the element size to be more refined in our channel then gradually increase to the domain extents.



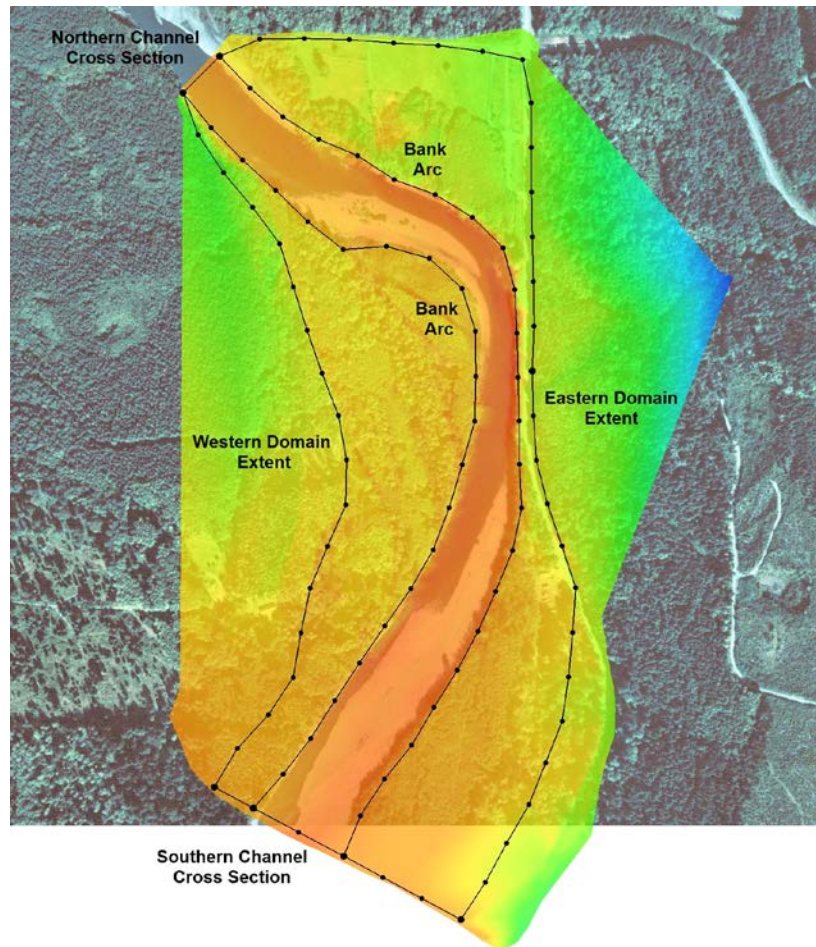



Figure : Feature Arcs to Redistribute

### 2.3.1 Redistribute the Channel Arcs

1. Select the arcs defining the East and West Banks of the channel as well as the Northern Channel Cross Section arc and the Southern Channel Cross Section arc (). To select multiple arcs hold down the shift key while using the *Select Feature Arc* tool .
2. Select Feature Object | Redistribute Vertices....
3. In the *Arc Redistribution* section select *Specified Spacing* from the *Specify* drop down menu. Set the *Spacing* to 40 ft.

4. Select *OK* to exit the *Redistribute Vertices* Dialog.

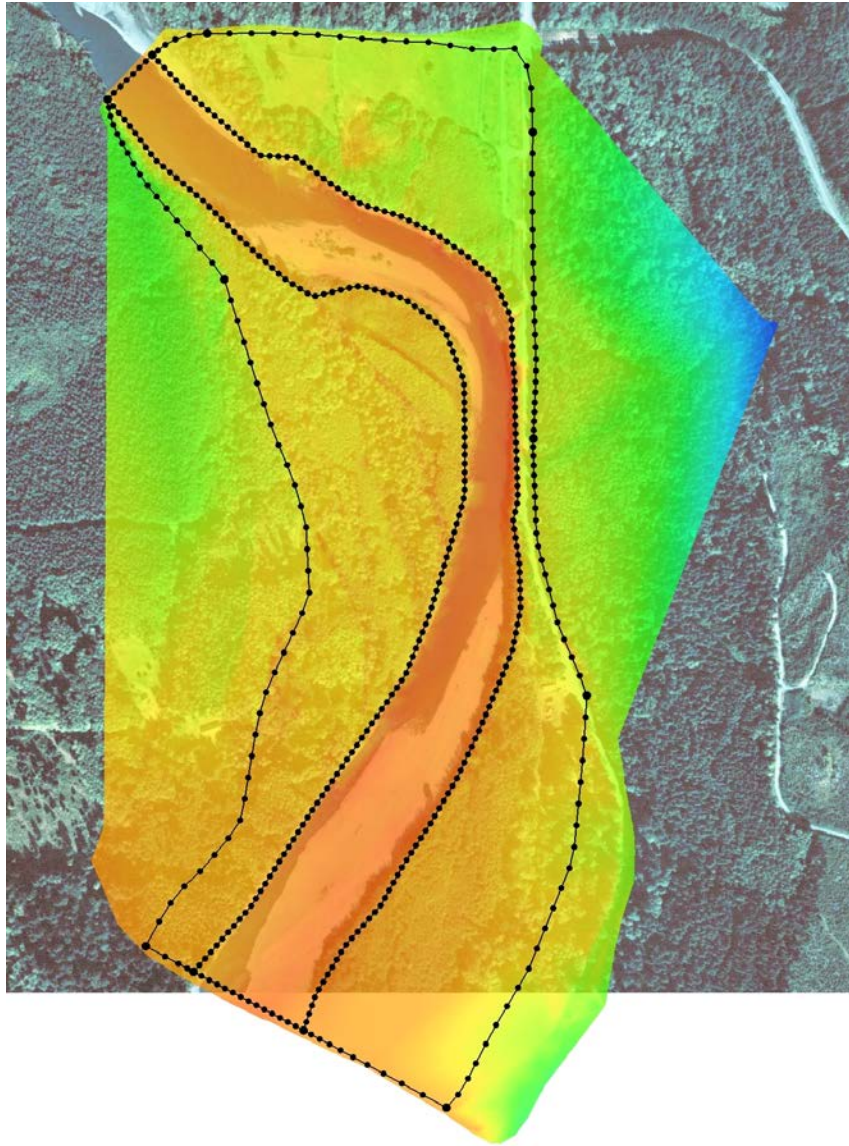
### 2.3.2 Redistribute the Domain extend arcs

1. Select the arcs defining the East and West Domain Extents ().
2. Select Feature Object | Redistribute Vertices....
3. In the *Arc Redistribution* section select *Specified Spacing* from the *Specify* drop down menu. Set the *Specified Spacing* 120 ft.
4. Select *OK* to exit the Redistribute Vertices Dialog.

## 2.4 Read in pre-processed map data

Now that you know how to define arcs and vertex spacing you are going to read in a file that has been created and the connection between the arcs have been smoothed with 40 ft spacing and 120 ft spacing. This will reduce the size change between neighboring elements that can cause instabilities in model runs. This file was created in a similar manner to the above steps using the *Min/Max Spacing* option as well as the *Specified Spacing* options and splitting a few of the features arcs previously created (changing a few of the vertices to nodes to form new arcs).

1. Select *File / Open* and find the *HohRiver\_Redistributed.map* file that is located with the other initial tutorial files.
2. Click *OK* to load this coverage.
3. A new coverage named “*Final Model*” should now be listed in the project explorer. Multiple coverages can be used in SMS but the rest of the tutorial will assume that you are working in the “*Final Model*” coverage. You may delete or turn off the original *Conceptual Model* coverage.



*Figure : Final Model Coverage*

### **3 Define Polygon Attributes**

---


#### **3.1 Polygon Materials**

1. Select *Feature Object / Build Polygons* to create three polygons from the defined arcs.

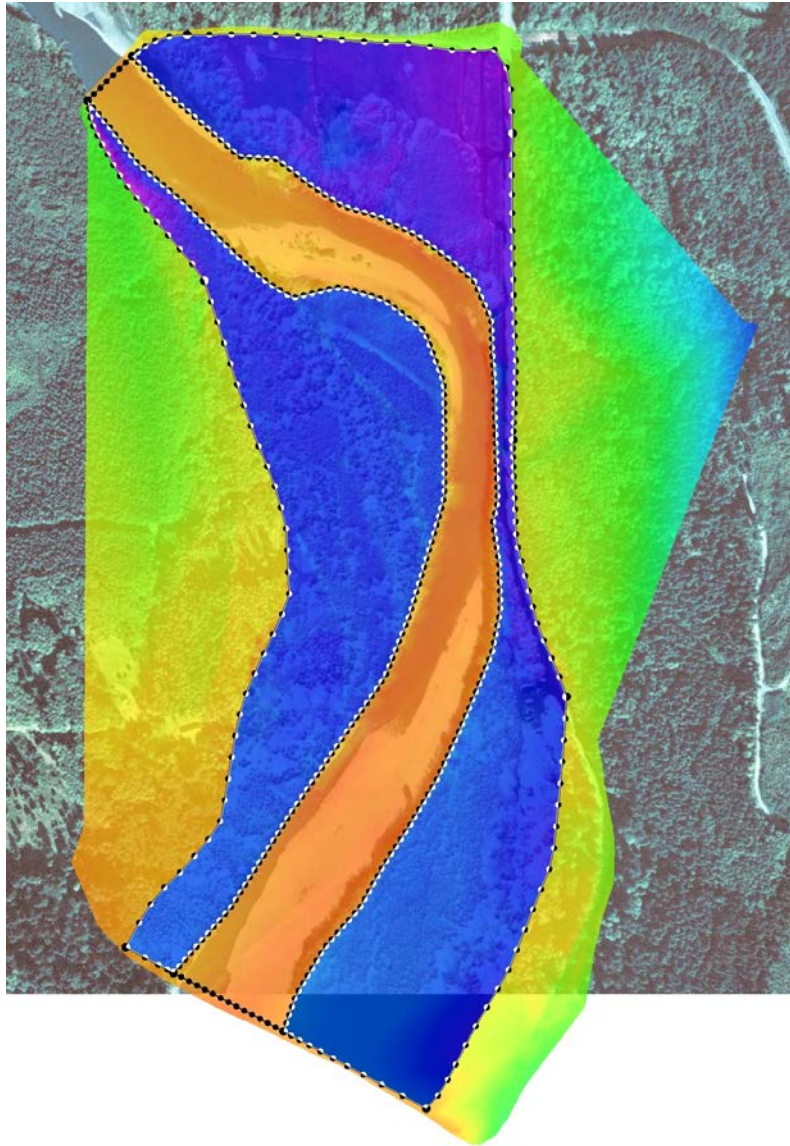
2. With polygons now created you can assign *Material Types* to them. First, you must specify what *Material Types* you wish to define then assign the *Material Type* to the different polygons.
3. Select *Edit / Material Data* to bring up the Material Data dialog.
4. By default there is a material named “*material 01*”. Select “*material 01*” and change the name to *Channel*.
5. Create a new material by selecting the *New* button and name this material *Floodplain*.
6. 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.

### 3.2 Defining Materials

RiverFLO-2D allows you to vary the Manning’s n value spatially. You have now defined two material types to assign to our polygons. Further in this tutorial you will assign Manning’s n values to these materials.

1. Select the Select Feature Polygon tool .
2. Select the floodplain polygons shown in below by holding the *shift* key and clicking within each of the polygons.
3. Select *Feature Object | Attributes...* to bring up the 2D Mesh Multiple Polygon Properties Dialog.
4. Toggle the *Mesh Type* and select *Paving* and leave the *Bias* at 1.00.





*Figure : Floodplain Polygons Selected*

SMS allows for different polygons to use different mesh types. RiverFLO-2D supports triangular mesh elements, so you will not be using the patch mesh type which fills the polygons with topologic rectangles. The scalar paving density method is a more advanced method than the normal paving option. It allows the user to define a dataset that depicts the node spacing spatially across the model domain. In this example you will select the paving mesh type and SMS will vary the element sizing linearly between the

vertex spacing on our previously created feature arcs. For more information regarding the different mesh types select the help button at the bottom left corner of the polygon attributes dialog and search for “Mesh Generation”.

1. Toggle the *Bathymetry type* option and select *Scatter Set* from the drop down menu.
2. Click the “*Scatter Options...*” button. In the *Scatter Set To Interpolate From* section, select the “Z” dataset and leave all other options at the default values. These options control what dataset to interpolate if we have multiple datasets loaded into SMS and what interpolation method will be used.
3. Select *Ok* to exit the Interpolation dialog.
4. Toggle on the *Material section*. Select the *Floodplain* option from the drop down menu
5. Click *OK* to exit.
6. Follow steps 1-5 with the center channel polygon and select *Channel* as the *material type* instead of *Floodplain* in Step 8. The dialog will have a *Preview Mesh* section and look slightly different since you have only one polygon selected but the options will be the same.
7. Select *OK* to exit any dialogs after setting polygon attributes.

## 4 RiverFLO-2D Boundary Conditions


---

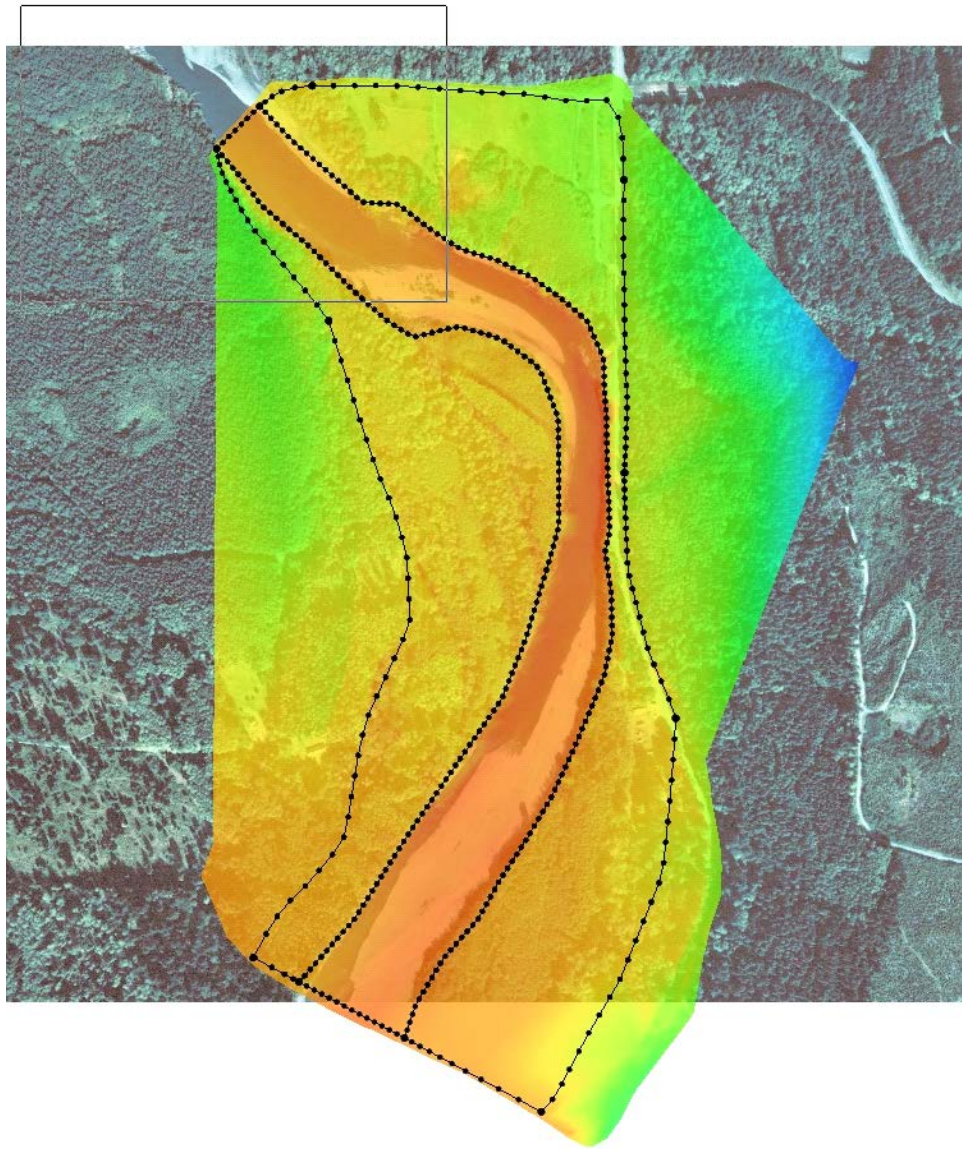
Boundary conditions force the model with certain hydrodynamic conditions. For this model, a flow discharge boundary condition will be specified at the upstream boundary and water surface elevation at the downstream.

### 4.1 Feature Arc Attributes

RiverFLO-2D Boundary conditions are assigned to the model by changing the feature arc attributes in the Map module. This section will describe how to assign different boundary conditions that will later be transfer automatically to the unstructured mesh.


#### 4.1.1 Upstream Boundary Condition

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



*Figure : Upstream Zoomed Area*



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

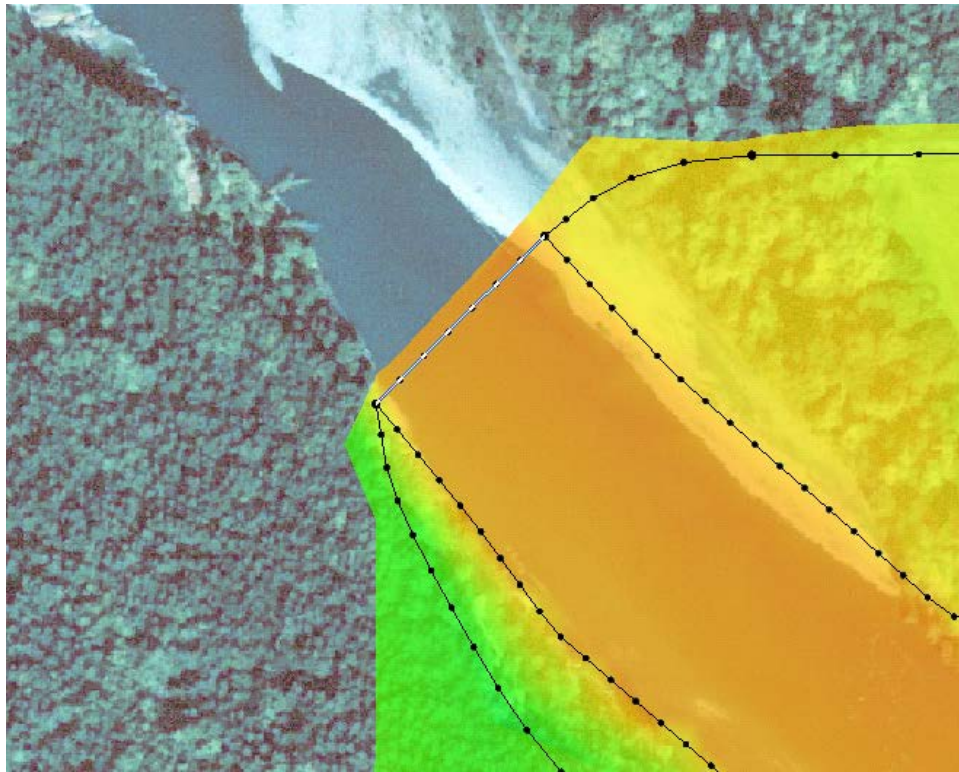


Figure : Upstream Node String

4. With the arc selected go to the *Feature Objects Menu* and choose *Attributes...* option.
5. In this dialog check the *Boundary Conditions* option and click on the *Options* button.
6. This will bring up the RiverFLO-2D Nodestring Boundary conditions dialog. Check the box next to the Exterior Boundary Condition and make sure that Discharge is selected as the Steady State BC Type.
7. In the *Discharge* field enter a value of 67116 cfs.
8. Click *OK* twice to exit out and get back to the main display.



### 4.1.2 Downstream Boundary Condition

The same process applies to the downstream boundary condition but for this tutorial the downstream boundary condition will be assigned directly to the mesh to demonstrate how to do this in SMS. This process is outlined in the next section.

## 5 Create Unstructured Mesh

---

### 5.1 Create Mesh

With the meshing parameters set, and the upstream boundary condition defined the model is ready to convert to a finite element mesh for RiverFLO-2D and then we will define the downstream Boundary condition directly to the mesh.

1. Select the “*Final Model*” coverage to make it active and to deselect any polygons that might be selected.
2. Select the Feature Objects | Map -> 2D Mesh menu item.
3. Click *OK* in the *2D Mesh Options* dialog leaving the default settings.

A finite element mesh with triangular elements 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:

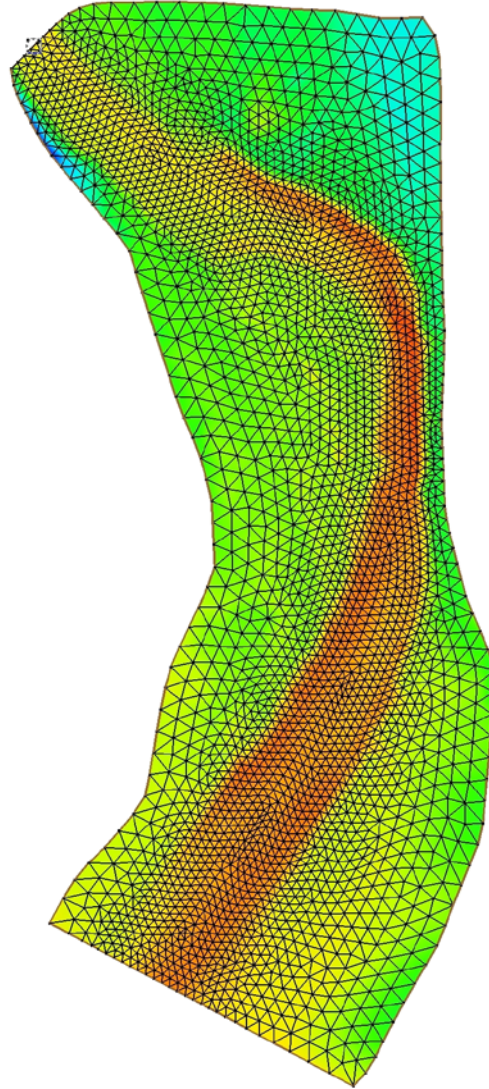
1. In the *Project Explorer*, uncheck the box next to the *HohRiver* image, the scatter set *BedElevations* and the map coverage(s).

### 5.2 Mesh Display Options

You will want to adjust our mesh display settings to see the elevations contoured.

1. Select *Display / Display Options*.
2. Select *2D Mesh* in the list on the left.
3. Toggle on *Contours*.
4. Under the *Contours* tab, change the *Contour Method* to *Color Fill*.
5. Click *OK* to exit the *Display Options* dialog.



The resulting image should appear similar to Figure 71 below.



*Figure : Mesh with Elevation Contours*


In SMS you can also visualize our channel in 3D.

1. Select *Display / Display Options*.
2. Select *General* in the list on the left.

3. Toggle off *Auto z-mag*.
4. Change the *Z magnification* to 10. This will help us visualize the elevation data since the range of values is quite different in the vertical direction than that in the horizontal direction.
5. Click *OK* to exit the *Display Options* dialog.
6. Select the *Rotate* tool  from the *Static Toolbar*.
7. Left click in the *Graphics Window* and drag to rotate the grid in 3D. You may also use the wheel of your mouse to zoom in and out.
8. You can get back to plan view by selecting the *Plan View Icon*  in the *Display Toolbar*.

### 5.3 Assigning Downstream Boundary Condition

Previously in our model we defined the upstream discharge as a boundary condition. This process was done independently of the mesh and SMS then automatically mapped the boundary condition over when the mesh was generated. This section describes how to assign a boundary condition directly to your mesh by creating a nodestrings.

1. In the *Mesh Module* select the *Create Nodestring tool* .
2. Select the corner downstream node and begin selecting the nodes along the downstream boundary. Double click on the last node to create the nodestring as shown below.

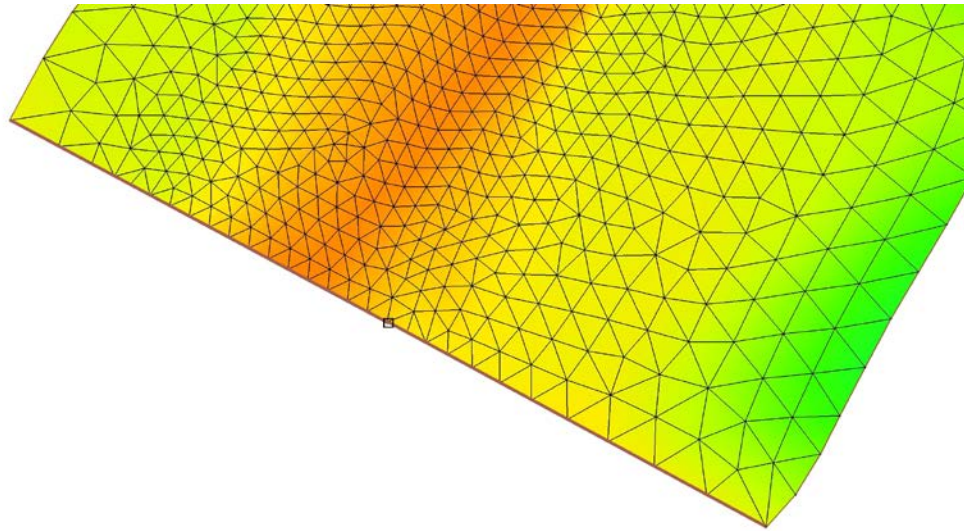



Figure : Downstream Nodestring

3. Select the *nodestring*  then under the *RiverFLO-2D* menu select the *Assign BC...* option.
4. Check the box next to the Exterior Boundary Condition and select Water Surface Elevation (WSE) as the Steady State BC Type.
5. In the *Water Surface Elevation* field enter a value of 166.07 ft.
6. Click *OK* to exit out and get back to the main display.

For display purposes SMS has arrows indicating the direction of flow when the nodestring is selected. If these arrows are pointed in the wrong direction the nodestring may be reversed by right clicking on the string and selecting the *Reverse Direction Option*.

## 5.4 RiverFLO-2D Components

In RiverFLO-2D there are different supported components that can be specified by selecting the nodestring or element and assigning a Component or Boundary Condition to



them. These options are not used in this tutorial but are available for other simulations. Here is an overview of the different options:

### **Nodestring**

Exterior Boundary Condition: Allows you to specify one of the supported *exterior boundary conditions*. See the RiverFLO-2D manual for more details for the different boundary condition types. This is normally done through the model but can be assigned directly to a nodestring.

Interior Boundary Condition: Allows you to specify a stage-discharge rating table along an internal line in your model. More details are in the RiverFLO-2D documentation

Weir: A weir may be defined across a nodestring to have RiverFLO-2D use the weir equation to calculate the flow from the upstream elements to the downstream.

### **Element**

Pier: Pier locations at the centroid of the element may be specified and RiverFLO-2D will simulate them using the drag equation. You may also simulate a pier by representing it with a polygon and turning the meshing option off in the polygon attributes in your model.

Source/Sink: The user may provide a hydrograph to simulate a point inflow or outflow. The Source/Sink location is the centroid of the element it is assigned to.

Culvert Inlet: A culvert can be defined by selecting an inlet and outlet element and specifying the culvert parameters in the Culvert Inlet dialog. The culvert parameters are then set depending on the culvert component type option.

Culvert Outlet: The outlet location of the culvert. The only parameter in this component is the name of the culvert specified in the Culvert Inlet. This allows you to specify multiple culverts. All other parameters are set in the inlet culvert dialog.

## **6 Material Properties**

---

### **6.1 Assigning Material Properties**

Each element of the mesh is assigned a material type. Each material type includes a value for Manning's roughness coefficient. These material properties must be changed for this analysis. We assign a value to each material type then SMS automatically assigns this value to the elements:

1. Select *RiverFLO-2D / Material Properties*.
2. In the *RiverFLO-2D Material Properties* dialog, highlight the material Channel.
3. Under the *Control Data* tab, enter a value of 0.035 for the *Manning's N*.

4. Highlight the material *Floodplain*.
5. Under the *Control Data* tab, enter a value of 0.040 for the *Manning's N*.
6. Click *OK*.

The material properties have now been properly defined.

Note: The material zones can be displayed by opening the *Display Options* dialog and turning on the *Materials* option under the *2D Mesh* tab.

## 7 RiverFLO-2D Control Data

---

### 7.1 Assigning RiverFLO-2D Parameters

Now that you have generated the unstructured mesh you need to set up our RiverFLO-2D control data and model parameters.

1. Select the *Mesh Data* in the Project explorer to make the *Mesh Module* the active module.
2. Select RiverFLO-2D Menu | Global Parameters...

This is the dialog in which you can set all the input control parameters for RiverFLO-2D. Here is an outline of each of the tabs in the RiverFLO-2D Global Parameters Dialog.

#### **Control Data**

Sets the model run control parameters like run time and time step duration. You may switch from a steady state model to a transient model and the associated parameters will be displayed and Boundary Condition options will update automatically. The initial conditions of your model can also be specified here.

#### **Sediment Control**

The sediment control tab displays one of the different sediment discharge formulas supported. As you select which method the required inputs are displayed.

#### **Rainfall/Evaporation**

In this dialog time varying rainfall and evaporation data may be defined. The RiverFLO-2D model assumes that this rainfall or evaporation is constant over your entire model.

## Graphic Output Options

Contains the parameters that can be set for the graphical output plots that are generated while RiverFLO-2D is running. These plots are useful to view potential problems so that you do not have to wait until the model converges to see any major issues.

## Component Controls

Allows you to turn on and off the different components that you have specified. Instead of deleting, for example, a set of piers you can simply turn the pier flag on and off to run the simulation with and without the effects of the piers.

1. Set the following parameters in the *Control Data tab* of the *RiverFLO-2D Global Parameters* dialog:
  - *Simulation Time*: 1 hour
  - *Output Interval*: 0.2 hours
  - *Variable Timestep*: On
  - *Variable Timestep Multiplier*: 1.0
  - *Simulation Type*: Steady State
  - *Initial Conditions*: Defined as Dataset (RBK "Define Initial WSE as Dataset"?)
  - *XMDF File Name*: InitialConditions.h5
  - *Mesh Dataset Name*: InitialWSE
  - *Wet-Dry Method*: B
  - All other parameters leave at their default
2. Select *OK* to end setting the *RiverFLO-2D Global Parameters*.

## 7.2 Assigning Initial Water Surface Elevation

In the *Control Data* tab the *Initial Conditions* was set as *Define Initial WSE as Dataset* option. There are two parameters, initial WSE and maximum sediment erosion depth, that can be spatially varied throughout your model domain and defined as a dataset from SMS. In this tutorial we will create a dataset then export the dataset to be read into RiverFLO-2D as our initial water surface elevations.

1. Select the *elevation* mesh dataset under the *Mesh Data* tree to activate the Mesh Module.
2. Select the *Data Menu* and then select the *Data Calculator...* option.

This opens the Dataset Toolbox dialog. This tool has a variety of options for manipulating datasets within SMS. For this example we are going to take the elevation dataset and add a 5 foot to create the initial WSE elevation value at each mesh node.

3. Double click on the *elevation* dataset under the mesh tree in the middle *Data Sets* box. You will notice a variable has been added to the *Calculator* field.
4. Select the “+” button and type the number 5 to add 5 feet to the variable that is representing the *Mesh* elevation dataset.
5. Change the *Output dataset name* to InitialWSE



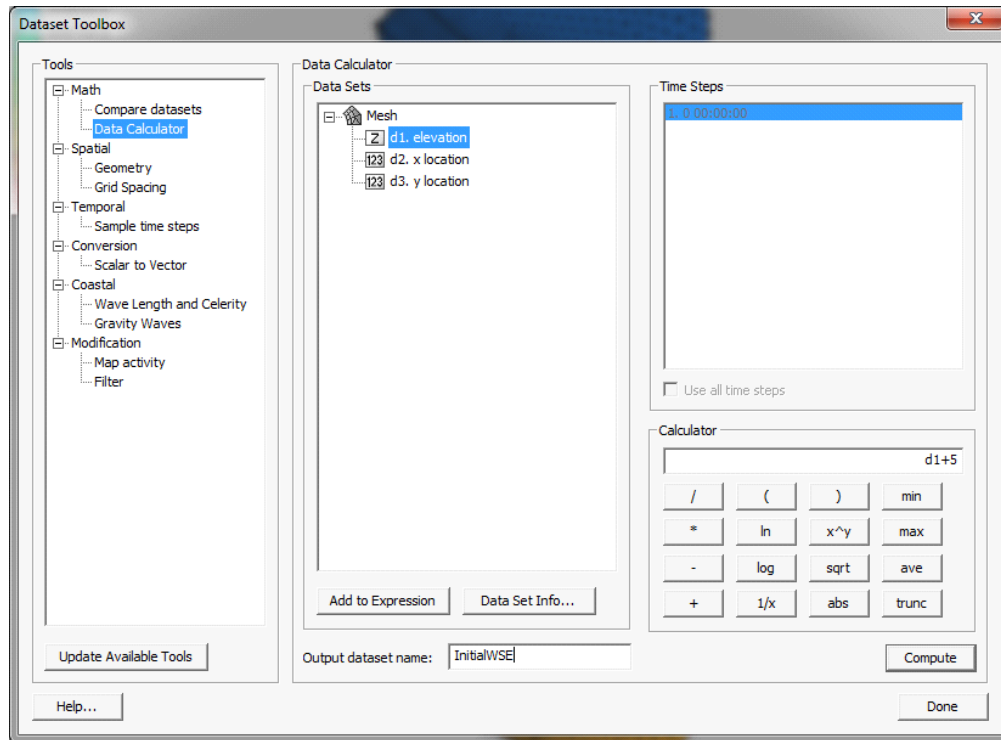



Figure : Dataset Calculator

6. Select *Compute* and SMS will create the new dataset.
7. Select *Done* to exit the dialog.
8. Right click on the *InitialWSE* dataset under the Mesh Data tree and select *Export...* to export the dataset.
9. Select XMDF file as the file type.
10. Select the *Folder* icon  and select the directory where your project is saved and change the name to *InitialConditions.h5*.

The name of the dataset and file exported must match exactly the text fields specified in the *Control Data* tab for RiverFLO-2D to recognize this dataset as the *Initial WSE*.

## 8 Running RiverFLO-2D

---

### 8.1 Run RiverFLO-2D

The Generic model interface allows any model to be used with SMS. To run the model you need to specify the directory where the RiverFLO-2D model executable is located

1. Select the *Edit / Preferences...* at the top of the SMS window.
2. Select the *File Locations* Tab
3. Scroll down in the *Model Executable* section till you see the *Generic* section.
4. Click the *BROWSE* button and find the directory that contains the RiverFLO-2D model executable and select the RiverFLO-2Dm3.exe executable
5. Select *OK* to exit.

At this point you can run the *SMS Model Checker* to see if there are any gross errors in our model setup.

6. In the *Mesh Module* select the *RiverFLO-2D* menu and select the *Check Mesh* option.
7. Read through any errors and try to resolve them.
8. Click File then Save As and name the project HohRiverFinal.sms.
9. Select the RiverFLO-2D menu then Run RiverFLO-2D.

SMS will save the .2dm file that contains the input parameters and mesh geometry to run RiverFLO-2D. After the model converges the model output will be in the XMDF format which can then be read into SMS and the results can be viewed. If you do not wish to wait for the model to finish you may read in the SMS project file from the output folder in the directory of the initial files. This contains a completed solution of the tutorial.



## 9 Viewing RiverFLO-2D Output

---

First you need to read in the XMDF output that RiverFLO-2D has generated.

1. Select *File / Open*.


2. Find the *HohRiver.hdf5* file located in the directory where your project is saved.
3. Click the *Open* button.

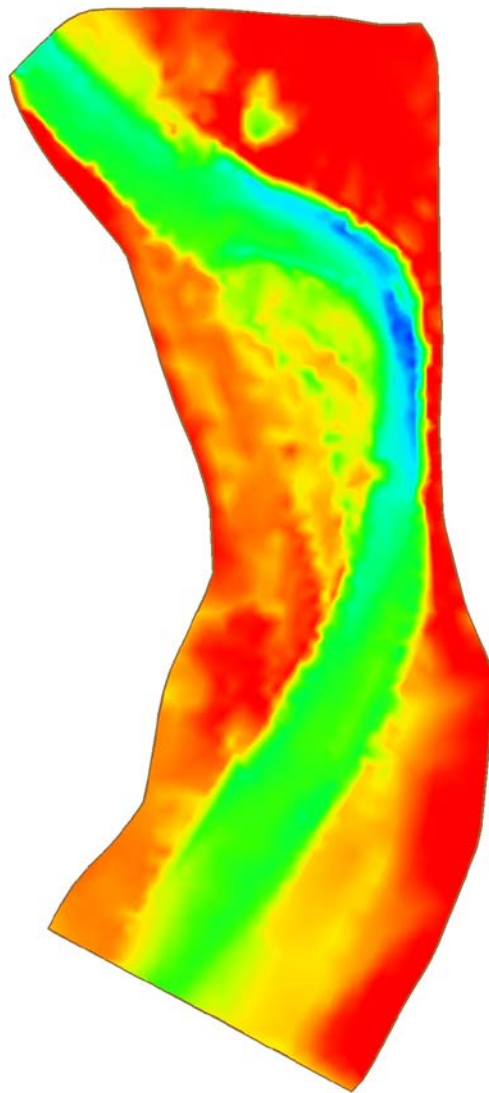
The RiverFLO-2D model output will be read into SMS as Mesh Datasets. Once the RiverFLO-2D output file has been imported, the user must decide on how to view the data. The Project Explorer may be used to select the desired Scalar and Vector output datasets. Scalar data sets are identified by the scalar icon  and the vector datasets have the vector icon  identifying them.

4. Activate the *Depth* output scalar dataset by selecting it in the *Project Explorer*.

## 9.1 Scalar Dataset Options

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

1. Select the *Display Options*  macro in the *Display Toolbar*.
2. In the *2D Mesh* tab, click the *All off* button to turn off current display options.
3. Turn on the Contours, and Mesh boundary.
4. Under the Contours Options tab, change the Contour Method to Color Fill.
5. For the *Number* of contours, enter 25.
6. Click *OK* to exit the dialog box, and SMS will redraw the screen similar to below.




*Figure : Depth Dataset Contoured*

## **9.2 Vector Dataset Options**


You can display velocity vectors several different ways. You will first view them displayed at each node, and then on a normalized grid.

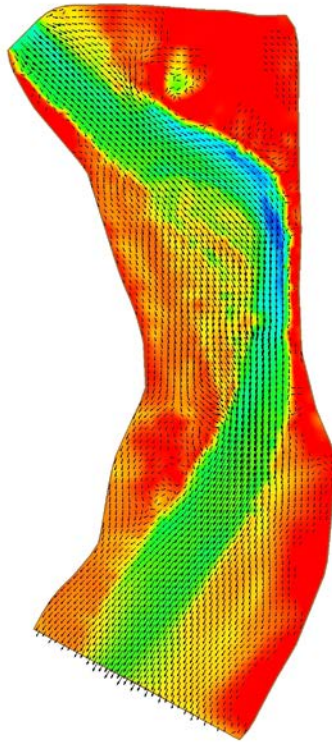


### 9.2.1 Vectors at Each Node

1. Activate the *Velocity output vector dataset* by selecting it 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 a value of 5 in the *Z-offset* and click the *OK* button. The *Z-offset* raises the origin of the vectors so that they are completely visible. Sometimes the vectors can display below the contours, this option corrects that.

### 9.2.2 Vectors on a Normalized Grid

1. Open the *Display Options*  and go to the *Vectors* tab.
2. Under *Vector Display Placement and Filter*, change the *Display* to the *on a grid* option.
3. For both the *X Spacing Pixels* and *Y Spacing Pixels*, enter a value of 15 and click the *OK* button.



*Figure : Vectors Normalized on a Grid*

This method of displaying vectors is useful when displaying areas with both coarse and refined areas. With the vectors displayed on a grid as you zoom in and out the resolution of your vectors displayed either increases or decreases to maintain the same pixel spacing.

### **9.3 Film Loop Visualization**

In addition to single time steps of contours and vectors, animations can be generated and saved. SMS enables the user to generate and save animations by using the Film Loop. To create a film loop of the RiverFLO-2D analysis:

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

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

SMS now starts 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.

You may continue to experiment with the film loop features if you desire. Click the Close button when finished. The film loop has been saved as sms.avi.

## **10 Conclusion**

---

This concludes the RiverFLO-2D tutorial.