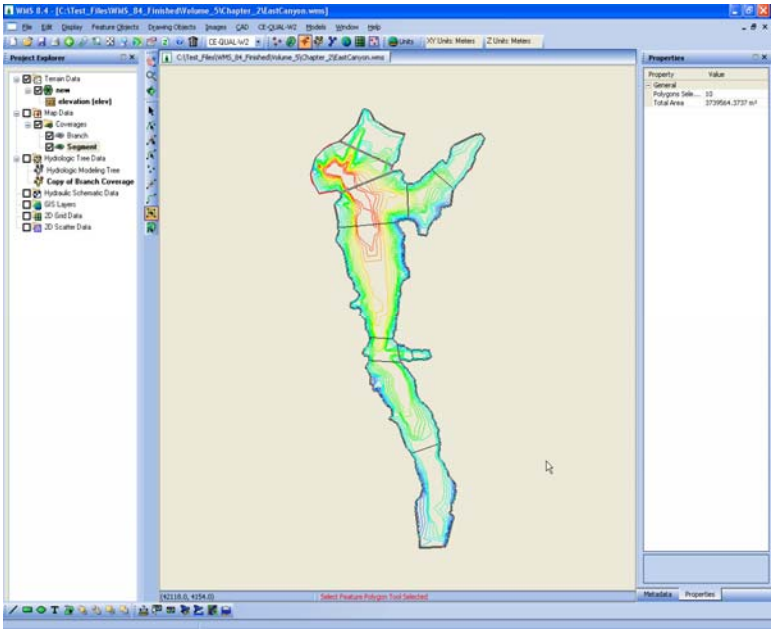
WMS

v. 9.0

WMS 9.0 Tutorial

Water Quality Modeling – CE-QUAL-W2 Interface

Learn how to setup a CE-QUAL-W2 model using the WMS interface



Objectives

In this tutorial, you setup a CE-QUAL-W2 model and its bathymetry data for a reservoir. Setting up CE-QUAL-W2 bathymetry data is one of the most time consuming tasks of setting up a model and involves dividing the reservoir into segments, branches, and layers and determining the geometric properties of each segment, branch, and layer. In addition to setting up bathymetric data for the reservoir, you will also setup water quality data and export the files necessary to run CE-QUAL-W2.

Prerequisite Tutorials

- Introduction – Basic Feature Objects
- Editing Elevations – Using TINs

Required Components

- Data
- Map
- 2D Grid
- CE-QUAL

Time

- 30-60 minutes

AQUAVEO™

1 Contents

1 Contents 2
2 Introduction 2
3 Objectives 2
4 Importing and Reprojecting the Data 3
5 Determining Reservoir Extent 3
5.1 Case 1: Automatic Boundary Polygon Generation 4
5.2 Case 2: Boundary of Reservoir to Polygon 5
6 Change Contour Display Options 5
7 Create Branch Coverage and Branches 5
7.1 Create Branch Coverage 5
7.2 Create Branches 6
8 Create Segment Coverage and Segments 7
8.1 Create Segment Coverage 7
8.2 Create Segments 7
9 System Modeling 8
9.1 Initializing CE-QUAL-W2 Simulation 9
9.2 Branch Identification 10
9.3 Segment Numbering 10
9.4 Editing Segment Properties 11
10 Editing an Existing CE-QUAL-W2 Model (Control File) 12
10.1 Time Step Job Control 13
10.2 Water Body Job Control 13
10.3 Editing Segments 14
10.4 Structures 14
10.5 Constituent Control 16
10.6 Kinetic Coefficients 16
11 Saving CE-QUAL-W2 Input Files 17

2 Introduction

CE-QUAL-W2 is a 2D laterally averaged hydrodynamic and water quality model. It can model vertical variance, eutrophication, seasonal turnover, algal blooms, etc. if incoming pollutants are known.

This exercise will walk you through the CE-QUAL-W2 WMS interface. It uses a previously generated TIN and creates CE-QUAL-W2 input files.

3 Objectives

In this exercise you will get acquainted with the CE-QUAL-W2 interface in WMS. This includes the following:

1. Determining reservoir extent
2. Creating branches
3. Creating segments
4. System modeling

5. Initializing CE-QUAL-W2 simulation
6. Identifying branches
7. Mapping segments to branches
8. Editing branch/segment properties
9. Saving CE-QUAL-W2 control input file
10. Saving bathymetry as CE-QUAL-W2 input file

4 Importing and Reprojecting the Data

The following steps will teach you how to import data and make sure they are in the proper coordinate system.

1. Close all instances of WMS
2. Open WMS
3. Select ***File | Open***
4. Locate the ***cequal*** folder in your tutorial files. If you have used default installation settings in WMS, the tutorial files will be located in ***\My documents\WMS 9.0\Tutorials\.***
5. Open the file named *"EastCanyon.tin"*
6. Right-click on *New* under *Terrain Data* and select ***Display Options***
7. On the TIN Data tab, deselect Locked Vertices, Unlocked Vertices, and Triangles
8. Select OK

CE-QUAL-W2 uses metric coordinates in all bathymetry calculations so it is advisable to convert to metric coordinates and enter all data in meters. If the user wishes to use feet for data entry then the coordinate conversion can be performed later with the same results. A prompt in these instructions will appear later advising the user when to make the conversion.

9. Select ***Edit | Current Projection...***
10. In the Current Projection dialog, select Local projection and U.S. Survey Feet for both Horizontal and Vertical units.
11. Select OK

5 Determining Reservoir Extent

Now that the TIN has been imported, the boundary of the reservoir must be defined.

1. Right-click on *new* under *Terrain Data* in the Project Explorer and select ***Display Options***
2. On the TIN Data tab, ensure that the TIN Contours box is checked
3. Select the Options button to the right of TIN Contours

4. In the Contour Method section, choose Color Fill from the drop-down box
5. Toggle on Specify a range under Data Range
6. Deselect Fill below
7. Set the Min as 5500 and the max as 5720
8. In the Contour Interval section, choose Specified Values, then click on the Populate Values button
9. Exit out of all dialogs by pressing OK

The extent of the reservoir is displayed. The minimum (Min) value was chosen as an elevation lower than any elevation in the reservoir. Max is the maximum water surface elevation, or the elevation at any desired stage to be modeled. By selecting to color fill using a range, the areas that fall between Min and Max have been displayed.

Once completed, the reservoir should appear similar to Figure 5-1.

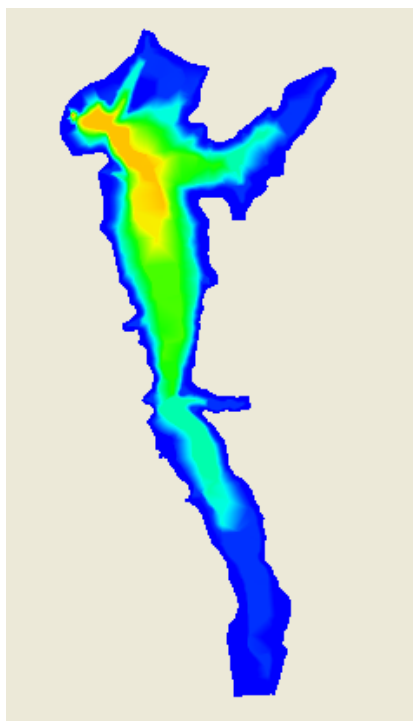


Figure 5-1: Reservoir Extent of East Canyon

There are two ways to generate the boundary of a reservoir. If you have refined your TIN such that the boundary of the TIN is the same general shape as the reservoir (see *Editing elevation – Using TINs* tutorial), then you can let WMS automatically generate the bounding polygon (Case 1). If not, you will need to manually create the bounding reservoir polygon (Case 2).

This tutorial's files follow Case 1; Case 2 is provided as a reference.

5.1 Case 1: Automatic Boundary Polygon Generation

1. Select the "new" tin from the data tree

2. Right-click on *new* under *Terrain Data* in the Project Explorer and select ***Convert | TIN Boundary -> Feature***

5.2 Case 2: Boundary of Reservoir to Polygon

1. Select the "Drainage" coverage from the data tree
2. Select the *Create Feature Arc* tool
3. Trace the outline of the reservoir at the specified water surface elevation by clicking at locations along the perimeter, making sure that the generated polygon stays within the boundary of the TIN

It is a good idea to create your polygon slightly larger than the boundary generated by the maximum water surface elevation. This way you ensure that the entire reservoir will be modeled. If you need to define the perimeter by defining separate arcs, double-click to end the arc. Make sure and start the new arc at the end of the old one however.

4. Right-click on the *General* coverage and select ***Build Polygon***
5. Select the *Select Feature Polygon* tool
6. Click somewhere within the reservoir. If all of the arcs have been connected correctly, the entire area will become highlighted.

6 Change Contour Display Options

For the following steps, we do not need to know the exact extent of the reservoir. Because color filled contours require more time for the image to regenerate, we will change the contour display options.

1. Select the *Contour Options* icon
2. Under Contour Interval, select Number of contours and enter 25
3. Under Contour Method, choose Normal Linear
4. Select OK

For WMS to process the data correctly, certain types of data must be grouped together as a coverage. In CE-QUAL-W2 modeling, two types of coverages must be created: the branch and the segment coverages.

7 Create Branch Coverage and Branches

Now you are ready to start creating branches and segments. You will first create the branch coverage.

7.1 Create Branch Coverage

1. In the Project Explorer, right-click the coverage labeled Drainage and select the ***Properties*** option
2. Change the Coverage type to CE-QUAL-W2 Branch

3. Set the elevation as 6000.0 ft, or any elevation above the maximum elevation of the TIN
4. Rename this coverage as “Branch”
5. Select OK

7.2 Create Branches

Now that the branch coverage has been created, the next step is to define the branches.

1. Select the *Create Feature Arc* tool
2. Use Figure 7-1 as a guide to create two feature arcs (branches) where indicated using the following two steps

Figure 7-1: Branch locations (shown by arrows)

3. Starting at one bank (at the edge of the polygon), click to begin the arc
4. Click on the other edge of the arc. If it has connected with the boundary polygon, it will automatically end the arc.

It is possible to create a curved boundary. Although this is not common practice, it can be done by simply creating more points along the arc.

5. These steps can be followed to define as many branches as desired, but the two shown in Figure 7-1 are sufficient
6. Right-click on the *Branch* coverage and select ***Build Polygon***
7. Select OK at the prompt
8. Choose the *Select Feature Polygon* tool

9. Click within each of the sections to make sure that polygons are correctly defining the branches. If they are, then the branch should become highlighted.

If the branches are not correctly identified when selected, then most likely the arcs created to define them did not snap to the boundary polygon or polygons have not been built. Using the zoom tool can help you identify these problems. You can also re-build your polygons after you make sure all arcs are correctly snapped. If problem persists, delete the created arcs, re-create arcs and re-build polygons.

8 Create Segment Coverage and Segments

We are now ready to start creating segments. We will start by creating a segment coverage. Because we have already defined the branches, we can use the branch boundaries to help define the segments.

8.1 Create Segment Coverage

1. In the Project Explorer, right-click the Branch coverage and select the ***Duplicate*** option
2. Right-click the duplicated coverage, Copy of Branch, and select the ***Properties*** option
3. Change the Coverage Type to CE-QUAL-W2 Segment
4. Make sure that the elevation is set to 6000.0, or any elevation above the high point of the TIN
5. Rename the coverage as “Segment”
6. Select OK

8.2 Create Segments

We will now create multiple subdivisions (segments) within the reservoir. While doing so, it is unnecessary to create segments where branches were defined previously.

1. Select the *Create Feature Arc* tool
2. Use Figure 8-1 as a guide to create feature arcs (segments) where indicated using the following two steps instructions (Remember that it is unnecessary to create segments where branches were defined previously)

Figure 8-1: Segment Locations

3. Starting at one bank (at the edge of the polygon at any node), click to begin the arc
4. Click on the other edge of the arc. If it has connected with the boundary polygon, it will automatically end the arc.

A few notes about segment creation:

- The arcs used to divide the segments should be roughly perpendicular to the general flow direction
- Only one segment can be present in the area where a branch intersects the main body (branches can only open into one segment)

5. Right-click on the *Segment* coverage and select ***Build Polygon***
6. Select OK at the prompt
7. Choose the *Select Feature Polygon* tool and click within each of the segments to make sure that polygons are correctly defining the segments
8. Select ***File | Save As***
9. Save your project as "EastCanyon"

9 System Modeling

After the segments and the branches have been created, it is a good idea to create a conceptual model of the system. Figure 9-1 shows a subdivided reservoir, and its accompanying model.

Important parts of the system model include the segments in each branch, the segment that a branch enters into, and the numbering. The general trend in the numbering is that the most upstream segment in each branch has the lowest value (beginning with the segments in the main branch). All segments in a branch should be numbered sequentially. In addition, the first segment in your model should be numbered 2. This is to allow for a dummy segment required by CE-QUAL-W2 to be created as segment 1. Dummy segments are also required at the end of each branch.

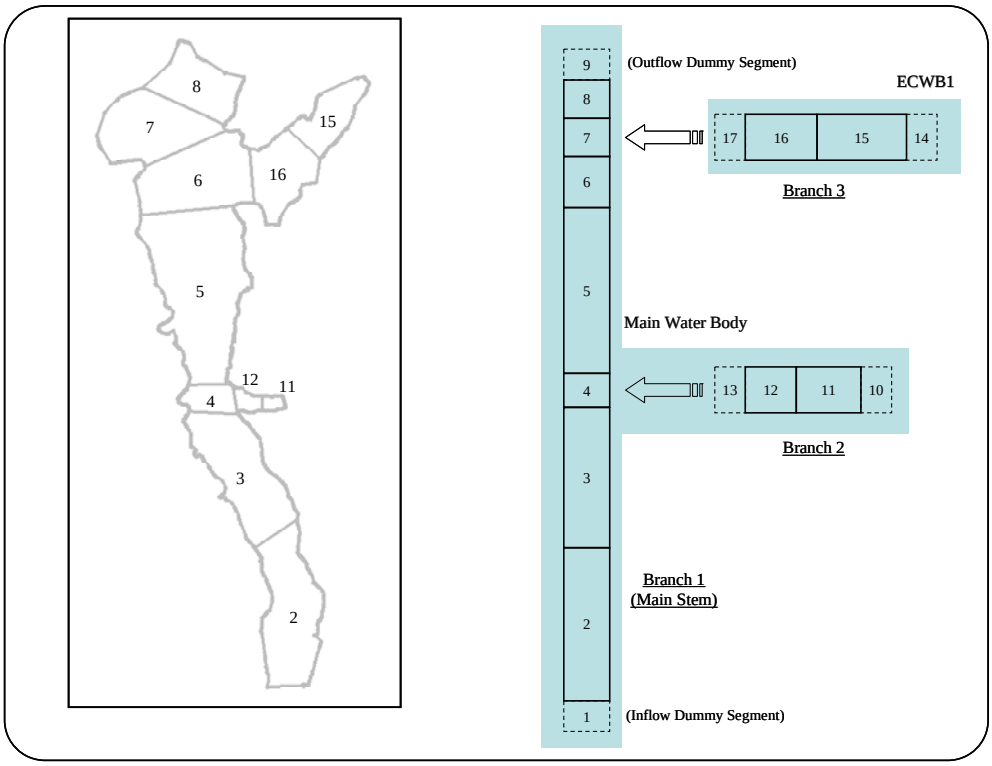


Figure 9-1: System Model

9.1 Initializing CE-QUAL-W2 Simulation

For WMS to generate input files required for CE-QUAL-W2, the interface must be initialized.

1. Select ***CE-QUAL-W2 | New Simulation***

The CE-QUAL-W2 interface is now initialized.

Before beginning branch identification the coordinates will need to be converted from U.S. survey feet to meters. This cannot be performed after obtaining the lengths of the branches since the measured lengths will not be converted along with the contours and polygons.

2. Select ***Edit | Reproject...***
3. In the New Projection section, change the Horizontal and Vertical units to Meters (make sure Local projection is selected).
4. Select OK

9.2 Branch Identification

Each branch must be given its corresponding identification as created in the System Modeling step.

1. Select ***CE-QUAL-W2 | Map Segments <=> Branches***
2. Select the *Select Feature Polygon* tool
3. Make Branch the active coverage by selecting it in the Project Explorer
4. Double-click on the main branch to bring up the Polygon Branch attributes
5. Select the Initialize branch properties button if it is not dimmed (If this button is dimmed, this means that this particular branch's properties are already initialized)
6. Check the box for Main stem
7. Select the *Get lengths and orientation of branch segments* tool
8. Select OK
9. Click just outside of the most upstream part of the branch (located at the southern end of the reservoir)

NOTE: The upstream segments of the different branches are labeled 2, 11, and 15 in Figure 9-1.

You will generally want the length of the branch to be equal to the average length of flow that will occur in a branch. To account for a curved flowpath, you can add as many points along the line as desired.

As you trace the line through the branch it is best to click at or in each segment to provide a better representation of the orientation of the branch and its segments. Avoid using one line to span multiple segments, rather click inside each segment for better results.

10. End the line by double-clicking just outside of the most downstream part of the branch

NOTE: The downstream segments of the different branches are labeled 8, 12, and 16 in Figure 9-1.

11. Select OK

This will give lengths and orientation to each segment in the branch and it also maps the segments to their respective branches.

12. Repeat these steps (4-5 and 7-11), selecting the two remaining branches, respectively, in the Polygon Branch Attributes dialog

9.3 Segment Numbering

Before continuing it is helpful to find the most upstream segment of each branch. In the segment coverage use the Select Feature Polygon tool to find the segment ID of the most upstream segment of each branch and make a note of the IDs.

1. Select the *Select Feature Polygon* tool
2. Double-click on the main branch

3. Select the segments tab in the polygon branch attributes dialog and highlight the most upstream segment ID

NOTE: To determine the most upstream segment, exit out of this dialog, highlight the segment coverage, and double-click on the most upstream segment in the desired branch, then you will be able to see the segment number in the Segment Attribute Dialog

4. Select the Make upstream segment button
5. Repeat steps 2-4 for the other branches
6. Select OK
7. Select ***CE-QUAL-W2 | Segment Numbering***
8. Select Yes

This numbers the segments starting with the upstream segment as segment 2 (segment 1 is a dummy segment that is inactive). This also designates the upstream and downstream segments of each branch.

9.4 Editing Segment Properties

Within each segment, many individual properties must be defined. These include the segment length, the layer properties, and the width computations. Figure 9-2 shows the different parameters required for the segment.

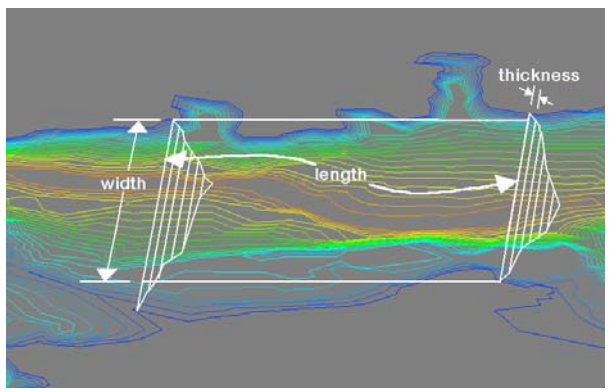


Figure 9-2: Segment bathymetry

1. Switch to the Segment coverage by selecting it in the Project Explorer
2. Highlight the entire reservoir by holding the SHIFT key and left-clicking each segment, or select ***Edit | Select All***
3. Select ***CE-QUAL-W2 | Layers***
4. Enter 1768 (meters) as the top elevation of the reservoir in the field labeled "Top elevation"

NOTE: You could also click on the Compute Top Elevation button to automatically compute the overall top elevation for the model.

5. Select the Compute Rating Curve button

The computation of the storage capacity curve may take some time. A visual display is generated showing the locations and elevations that are being modeled. Figure 6 shows the CE-QUAL-W2 layer editor where the storage capacity curve is shown.

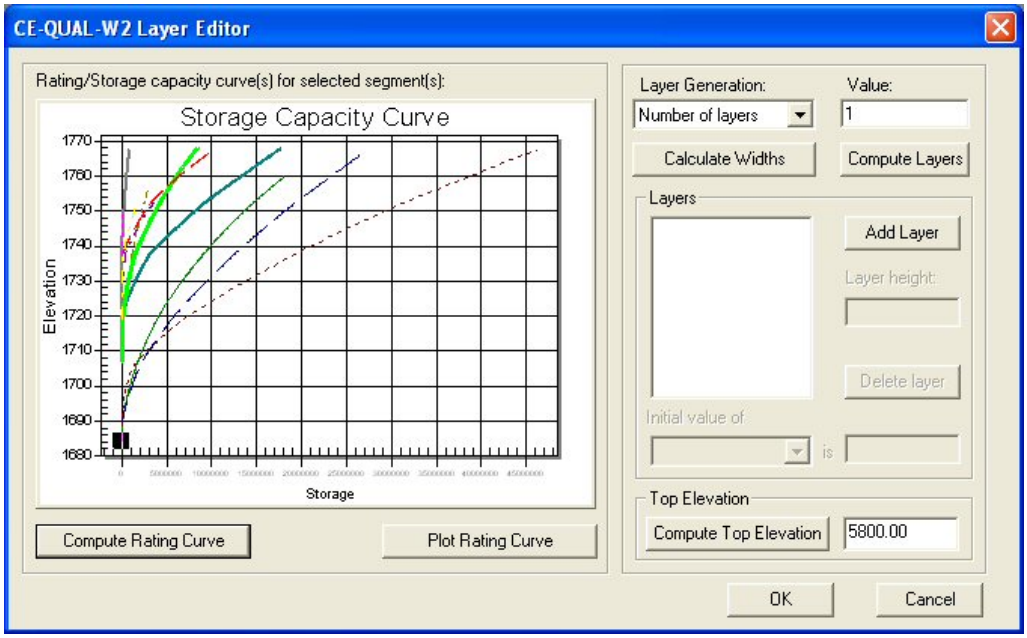


Figure 9-3: CE-QUAL-W2 Layer Editor

6. Select Layer Heights from the Layer Generation pulldown menu
7. Enter 10.0 in the Value field

Notice that you can also enter in a number of layers instead of a layer height. If the number of layers option is chosen, then the layer height will be calculated using the following equation:

H = (Max elevation on TIN – Min elevation on TIN) / Number of layers

If the layer height option is used, then all of the layers will be that specified thickness, except for the top layer, which will be a remainder of what is left.

8. Click on the Compute Layers and Layer Widths button
9. Exit out of the layer editor by selecting OK

WMS computes the widths of the segments using the layer height, length, and the volume computed from the storage capacity curve. In order for a layer to have a zero width, then its volume must be zero, meaning that the minimum elevation of the segment lies above the maximum elevation of the layer.

10 Editing an Existing CE-QUAL-W2 Model (Control File)

There are two main input files required by CE-QUAL-W2: bathymetry and control files. In this section, we will learn to generate/modify control file parameters.

10.1 Time Step Job Control

1. Select ***CE-QUAL-W2 | Time Step Job Control***
2. Enter "CE-QUAL-W2 exercise model" in the first line of the Title field
3. Enter "East Canyon" in the second line of the Title field
4. Select Modify from the Start Date/Time frame
5. Enter 1998 for the year, 2 for the month and 1 for the day. Leave the hours, minutes and seconds fields as default
6. Select OK
7. Similarly, enter 1999 for the year, 11 for the month and 30 for the end date/time
8. In the Time Step Intervals frame, click on Add interval
9. In the Time Step Intervals list box, click on the generated interval
10. Select Modify in the Selected Interval Parameters frame
11. In the Reference Time dialog, change the day to 9
12. Select OK
13. Modify the Maximum Timestep to be 100.0 and the Fraction of Timestep to be 0.9
14. In the General Timestep Parameters frame, modify the Maximum Output Dates to 100
15. Click OK to close the Time Step Job Control dialog

Now that we have set the time step parameters along with some other general model parameters, we are ready for more model-specific parameters.

10.2 Water Body Job Control

The new version of CE-QUAL-W2 uses the concept of Water Bodies. A water body can contain one or more branches. Any CE-QUAL-W2 model must have at least one water body. At the start of building a model, WMS will create one water body, Main Water Body, which has all branches assigned to it by default. Nevertheless, users can create water bodies, name them as they want, and switch branches back and forth between water bodies.

WMS realizes which segments lie within what branches. This helps WMS when assigning branches to the Main Water Body and automatically detects the location of each water body.

1. Select ***CE-QUAL-W2 | Water Body Job Control***
2. Select Main water body from the Current Water Bodies list box on the left (by doing this, all tabs in the dialog will be undimmed accordingly)
3. Choose the General tab

NOTE: The values shown for X and Y coordinates are the automatically detected locations of the first segment that lies in the Main Water Body. These values can be

manually edited to write geographic coordinates (required by CE-QUAL-W2), or users could use the Convert to Lat/Lon button.

4. Enter 41.5 for X-Coordinate (Latitude) and -118.0 for the Longitude (Y-Coordinate)
5. Click the Output tab
6. Leave everything as default and toggle on Screen Output, Profile Plot Output, and Snap Shot Output
7. Click on the Dates button for Screen Output
8. Click Add Date
9. Click on the added date
10. Click Edit
11. Change the year to 1998, the month to 5, and the day to 1
12. Click OK
13. Change Frequency to 10
14. Click OK
15. Do the same (steps 7-14) for Profile Plot Output and Snapshot Output
16. Click OK to close the Water Body Job control dialog

We are now ready to use the initial input parameters for CE-QUAL-W2. We will use the Water Body Job Control dialog later to enter constituent details and kinetic coefficients.

10.3 Editing Segments

In this section you will learn to edit segment parameters, define length, and set orientation and friction parameters.

1. Select ***CE-QUAL-W2 | Segments***
2. Click on the first segment on the list (Segment 2 from the Segment list box)

You will notice that this segment has a length and orientation already defined in the length of segment and segment orientation fields respectively. These values were defined when you used the measure tool to measure the branches in section 9.2. If segment length and orientation are not already defined, you can define these values using the measure tool in the segments dialog.

3. Enter 0.04 for the value of Bottom Friction
4. Enter this same value for Bottom Friction for all the segments in the model
5. Click OK to return back to your model in the graphics window

10.4 Structures

In this section you will learn to add/modify structures to your model. WMS will let you add one structure for each node in the branch coverage. However, if you have multiple

structures in the same place, you can simply add as many nodes as needed close to that location and assign a structure for every node.

1. Click on the Branch coverage in the Project Explorer. This will make it the active coverage
2. Create a new node as shown in the following figure using the *Create Feature Point* tool , as shown by the arrow in Figure 10-1

Figure 10-1: Insert Node

3. Click on the *Select Feature Point/Node* tool
4. Double-click on the node you just created
5. Click on Structure radio button in the Node Attributes dialog. This will activate/undim the structure group.
6. Click on Spillway…. This will open the Define Spillways dialog.
7. Enter 2 for the Spillway outflow segment, and 1676.4 for the spillway elevation
8. Enter 44, 1.5, 34 and 1 for a1, b1, a2 and b2, respectively
9. Toggle on the Compute dissolved gas check box
10. Choose spillway equation number 2
11. Enter 10.0, 110.0 and 10.0 for A, B, and C, respectively
12. Select DISTR from Spillway Flow Type combo box in the Upstream Spillway frame
13. Enter 1697.7, 1691.6, 3, 4 for Spillway Top Elevation, Spillway Bottom Elevation, Top Withdrawal Layer and Bottom Withdrawal Layer, respectively
14. Click OK to close the Define Spillway dialog
15. Click OK to close the Node Attributes dialog

Now you have defined the spillway at the specified location. You can also define other structures as needed.

10.5 Constituent Control

At some point in your process of your CE-QUAL-W2 model, you will need to modify/update your constituents and kinetic coefficients. In this section you will learn how this is accomplished in WMS.

1. Select ***CE-QUAL-W2 |Water Body Job Control***
2. Select Main Water Body from the Current Water Bodies list box on the left
3. Click on the Calculations tab
4. Turn on the Compute toggle in the Constituent computations frame
5. Click the Constituent Control button to bring up the Constituent Control dialog
6. In the Additional Constituents frame, select Algal Group from the Constituent type combo box, and leave Algae 1 as the default name
7. Click the Add Constituent button
8. Similarly, add another Algal group, Algae 2, and two Epiphyte groups: Epiphyte 1 and 2
9. Using the Control key, multi-select Algae, Algae 1, Algae 2, CBOD, Epiphyte 1 and Epiphyte 2
10. Toggle on the Active check box
11. Click OK. This should bring you back to the Water Body Job Control; Calculations tab. Do not close this dialog; we will need it for the next section.

CE-QUAL-W2 enables modelers to create their own version of five pre-specified constituent groups: algae, epiphyte, CBOD, generic, and inorganic suspended solids. This section described how to add these. In the next section, you will learn how to modify the constituent-specific kinetic coefficients.

10.6 Kinetic Coefficients

In this section you will learn how to edit algal and epiphyte group parameters.

1. Make sure the Calculations tab in the Water Body Job Control is open
2. Click on the Kinetic Coefficients button
3. Make sure that Algae tab is selected. Select Select All Algal Groups from the Algal group combo box
4. Change AT2 to 32.5
5. Select Algae 2 from the Algal group box. And change AT2 to 37.5
6. Select Algae 1 from the Algal group box. And change AT2 to 22.5

We now assume that all algal groups share the same coefficients except AT2, which is different for Algae 1 and 2 as specified above.

7. Select the Epiphyte tab and choose Select All Epiphyte from the Epiphyte group combo box

8. Change ESAT to be 55
9. Select Epiphyte 1 from the Epiphyte group combo box
10. Change ET1, ET2, EK3, EK4, EG, ER to be 10, 32, 0.75, 0.5, 7, 0.1 respectively

Similar to the algal groups, we assumed that all epiphyte groups share the same coefficients except ET1, ET2, EK3, EK4, EG, and ER, which are different for Epiphyte 1, as specified above.

11. Select OK twice

11 Saving CE-QUAL-W2 Input Files

In this section you will learn about the CE-QUAL-W2 input files that will be created by WMS.

WMS generates one control file for your model. However, it can create one or multiple bathymetry files depending on how many water bodies you have in your model. In other words, we can say that WMS will generate one bathymetry file for every water body in the model.

1. Select ***CE-QUAL-W2 | Save Simulation***
2. Find an appropriate folder and save the simulation there
3. For the control file name, make sure that the prefix is no longer than 8 characters long.

NOTE: you can only change the bathymetry file prefix to any name provided that is not more than 4 characters long. This is because WMS automatically adds four additional characters to that prefix. These are _wb1, _wb2 and so on for as many water bodies as you might have in your model.

In saving the simulation, two main files are created: the control file and the bathymetry file. One control file is generated for the whole model, however, there are multiple bathymetry files generated for each water body in the model.

