

---

## **CE-QUAL-W2 Interface**

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.

### **2.1 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

## 2.2 Importing Data and Coordinate Conversion

---

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 folder **C:\WMS80\tutorial\cequal**
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**


WMS coordinates are set to U.S. Survey Feet by default. This should be recognized every time a file is opened in WMS. If the file was saved using metric coordinates then this will need to be updated each time the file is opened. 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. Right-click on *Terrain Data* in the Project Explorer and select **Coordinate Conversion**
10. In the Coordinate Conversion dialog check the *Edit project coordinate system* box
11. If the current project units are in U.S. Survey Feet, exit out of this dialog by clicking **OK**. Otherwise, change the horizontal and vertical units to the appropriate system (U.S. Survey Feet) and then click **OK** to exit the dialog

## 2.3 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 *Contours* button to the right of *TIN Contours*
4. In the Contour Interval section, choose *Specified Values*, then click on the *Populate Values* button
5. Exit by pressing *OK*
6. In the Contour Method section, choose *Color Fill* from the drop-down box
7. Select *Specify a range* under Data Range
8. Deselect *Fill below*
9. Set the Min as **5500** and the max as **5720**
10. Exit out of both 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 2-1.

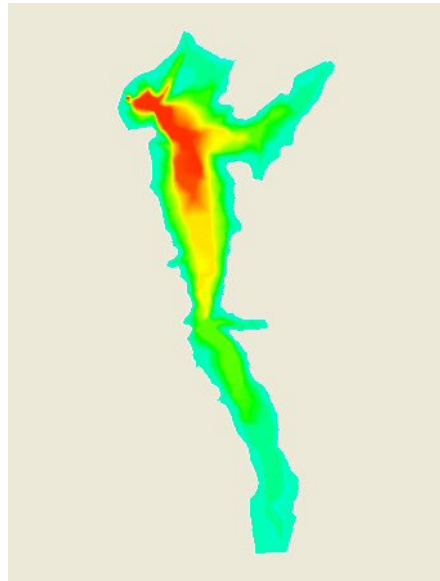


Figure 2-1: Reservoir Extent of East Canyon

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 Volume 1, Chapter 5), 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.

### 2.3.1 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**

### 2.3.2 Case 2: Boundary of Reservoir to Polygon


---



1. Select the “General” 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.

## 2.4 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 create: the branch and the segment coverages.

## 2.5 Create Branch Coverage and Branches

---

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

### 2.5.1 Create Branch Coverage

---

1. In the Project Explorer, right-click the coverage labeled *General* and select the **Properties** option
2. Change the Coverage type to *CE-QUAL-W2 Branch*
3. Set the elevation as **6000.0**, or any elevation above the maximum elevation of the TIN
4. Rename this coverage as “*Branch*”
5. Select *OK*

## 2.5.2 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 2-2 as a guide to create two feature arcs (branches) where indicated using the following two steps instructions




Figure 2-2: 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 2-2 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.

## 2.6 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.

### 2.6.1 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*

### 2.6.2 Create Segments

---

We will use 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 2-3 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 2-3: 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 prompts
  7. Choose the *Select Feature Polygon* tool  and click within each of the segments to make sure that polygons are correctly defining the branches



8. Select **File | Save As** 
9. Save your project as “EastCanyon”

## 2.7 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 2-4 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.

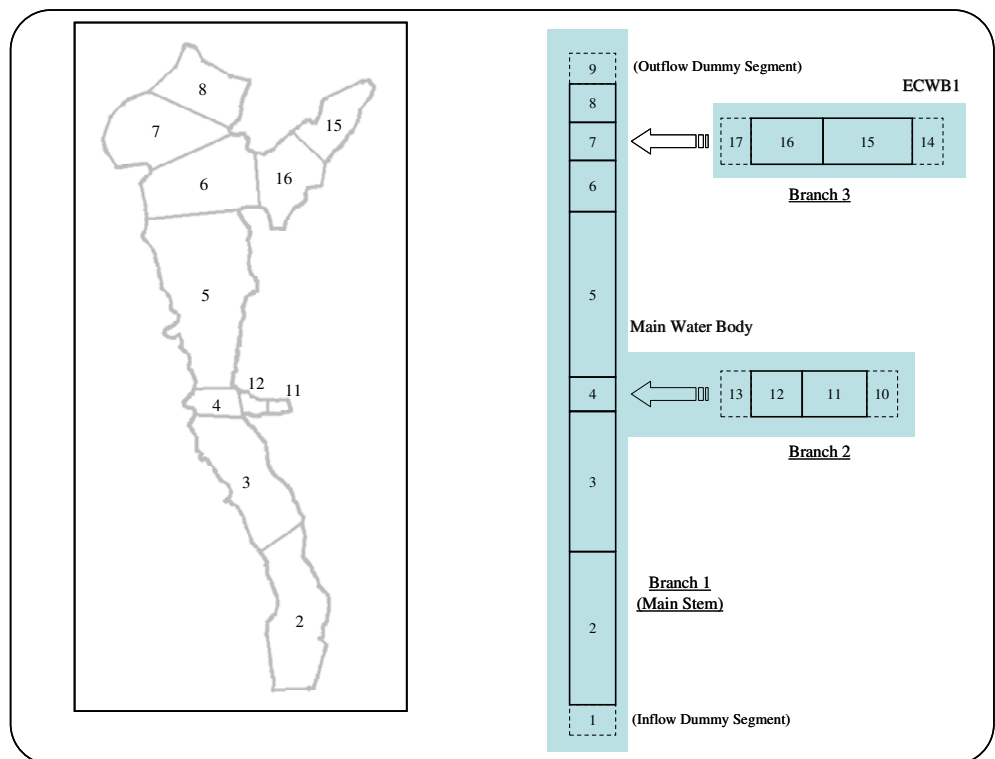


Figure 2-4: System Model

### 2.7.1 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 | Coordinate Conversion***
3. In the Convert To tab, change the Horizontal and Vertical units to *Meters*
4. Select *OK*

### 2.7.2 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. Ensure that the proper branch is selected from the list on the left
6. 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)
7. Check the box for *Main stem*
8. Select the *Length of Segment* tool 
9. Select *OK*
10. Click just outside of the most upstream part of the branch

NOTE: The upstream segments of the different branches are labeled 2, 11, and 15 in Figure 2-4.

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 at each segment for better results.

11. 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 2-4.

12. Select *OK*


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

13. Repeat these steps (4-6 and 8-11), selecting the two remaining branches, respectively, in the Polygon Branch Attributes dialog

### 2.7.3 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 know 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.

#### 2.7.4 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 2-5 shows the different parameters required for the segment.

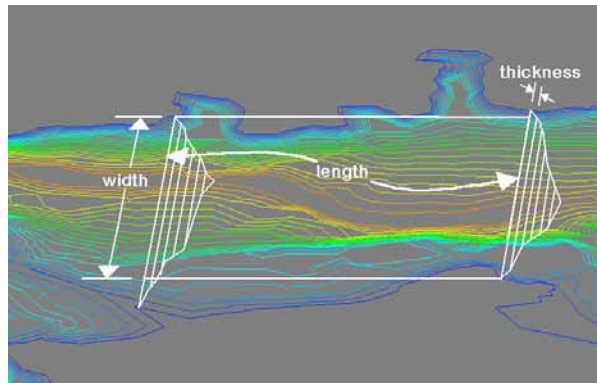


Figure 2-5: Segment bathymetry

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.

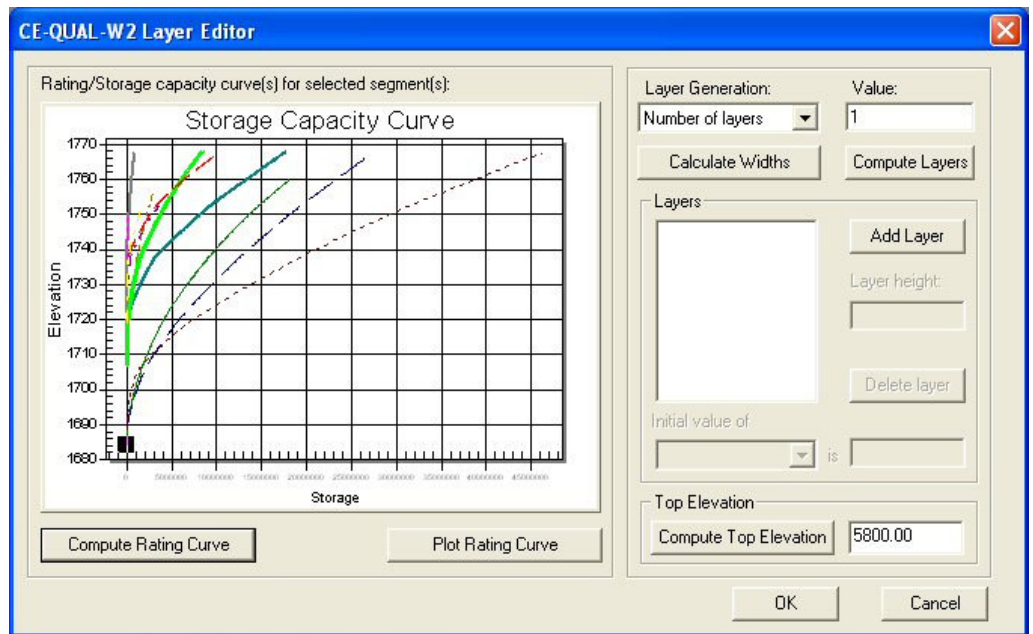


Figure 2-6: CE-QUAL-W2 Layer Editor

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:

$$H = (\text{Max elevation on TIN} - \text{Min elevation on TIN}) / \text{Number of layers}$$

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* button
9. Click on the *Calculate Widths* button. If a prompt telling you that “Some of the layers are below the minimum elevations in the storage capacity curve” appears, select *OK*.
10. 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.

## 2.8 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.

### 2.8.1 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, use *30 Nov. 1999* as an 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.

## 2.8.2 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. Choose the Assignment tab
6. Toggle *Add water body* on in the Additional Water Bodies frame
7. Enter “*ECWB1*” in the name field
8. Select the *Add Water Body* button

This will add ECWB1 to the Current Water Bodies list boxes (in the main dialog and in the Assignment tab).

9. Do the same and create a new water body called “*ECWB2*”
10. Select *ECWB1* in the Current Water Bodies list box in the Assignment tab.

Note the Branches in the selected Water Bodies list box. It displays “Currently, no branches are associated with ‘ECWB1’.”

11. Select *Main Water Body* in the Current Water Bodies list box in the Assignment tab. You will see a list of all the branches associated with the Main water body.
12. Use the CTRL key to multi-select branches 2 and 3 from the Branches in Selected Water Bodies list box
13. Click the *Remove from Selected Water Body* button. This will leave those branches unassigned and they will be listed in the Available Branches list box.
14. Select *ECWB1* in the Current Water Bodies list box in the Assignment tab
15. Select *Branch 2* in the Available Branches list box
16. Click *Add to Selected Water Body*. This will add the selected branch to the selected water body (ECWB1).
17. Make sure that *ECWB1* is selected in the Current Water Bodies frame in the Assignment tab
18. Click *Remove Water Body*. You will be prompted that removing a water body will re-allocate all its branches to the Main Water Body.
19. Click *OK*
20. Select *ECWB2* in the Current Water Bodies list box in the Assignment tab
21. Select *Branch 3* in the Available Branches list box
22. Click *Add to Selected Water Body*. This will add the selected branch to the selected water body (ECWB2)

This way we can switch branches back and forth between different water bodies. Notice that we can not delete the Main Water Body.

23. Click the Output tab
24. Leave everything as default and toggle on *Screen Output*, *Profile Plot Output*, and *Snap Shot Output*
25. Click on the *Dates* button for Screen Output
26. Click *Add Date*
27. Click on the added date



28. Click *Edit*
29. Change the year to **1998**, the month to **5**, and the day to **1**
30. Click *OK*
31. Change Frequency to **10**
32. Click *OK*
33. Do the same for *Profile Plot Output* and *Snapshot Output*
34. 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.


### 2.8.3 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 Segments list box on the left)

You will notice that this segment has a length and orientation already defined in the length of segment and segment orientation fields respectively. If segment length and orientation are not already defined, you can define them by following steps 3-5.

3. Click on the *Measure* tool . This will let you get the length and the orientation of the selected segment.
4. WMS will highlight the segment you just selected in the Segments list box to make it easier for you to trace it
5. Trace a line downstream that starts immediately upstream of the segment till immediately downstream the segment

NOTE: It is advisable to keep segments as straight as possible so that one line can be a good representation. However, in some situations, modelers will be confronted with the fact that they will have to merge segments into each other to reduce the overall number of segments which will correspondingly enhance run time for CE-QUAL-W2. In this case, trace the segment downstream and

WMS will get an orientation of a line that starts at the very upstream of the segment and continues to the very downstream of the segment.

6. Enter **0.04** for the value of Bottom Friction
7. Do the same for all the segments in the model
8. Select *Segment Vertical Profile*
9. Select *Global Vertical Profile Options*. This will bring up the CE-QUAL-W2 Layer Editor dialog.
10. Click on *Compute Top Elevation* in the bottom right corner of the dialog
11. Enter **5** in the Value field at the top right corner of the dialog. Make sure *Number of layers* is selected in the Layer Generation combo box
12. Click *Compute Layers*
13. Click *OK* three times to return back to your model in the graphics window

You have now successfully edited segments parameters. We will now edit the branches.

#### 2.8.4 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. Yet, if you have multiple structures in the same place, you can simply add as many nodes as needed and assign 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 2-7



Figure 2-7: 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 *Spillways*. 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 we have defined spillway at the specified location. You can also define other structure as needed and in the same manner.

Remember, you can create multiple structures in the same location by creating multiple adjacent nodes and assign structures to them.

### 2.8.5 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 (by doing this, all tabs in the dialog will be undimmed accordingly)
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 three Epiphyte groups: Epiphyte 1, 2, and 3
9. In the Current constituents list box, choose Epiphyte 3 and click *Delete Constituent* button. This should delete it from the list of constituents.
10. Using the Control key, multi-select Algae, Algae 1, Algae 2, CBOD, Epiphyte 1 and Epiphyte 2
11. Toggle on the Active check box
12. 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. We have discussed how to add/delete these. In the next

section, we will see how to modify their constituent-specific kinetic coefficients.

### 2.8.6 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

## 2.9 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.