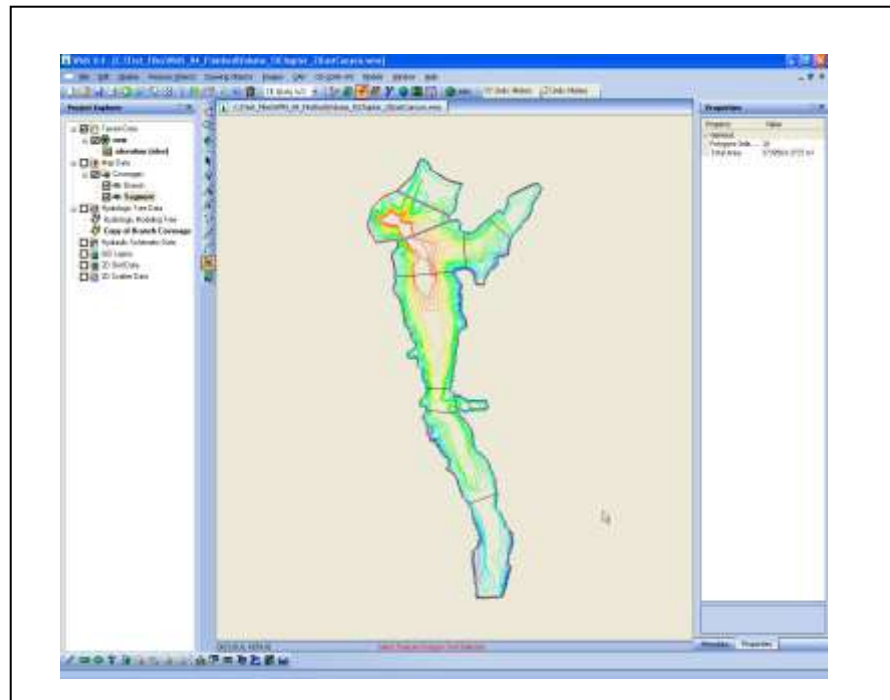*WMS 10.0 Tutorial*

Water Quality Modeling – CE-QUAL-W2 Interface

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

Objectives

In this tutorial, users 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, users 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   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 users through the CE-QUAL-W2 WMS interface. It uses a previously generated TIN and creates CE-QUAL-W2 input files.

2   Objectives

In this exercise users 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

© Aquaveo 2015

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

3   Importing and Reprojecting the Data

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

CE-QUAL-W2 uses metric units in all bathymetry calculations so it is advisable to enter all data in meters. The units used for the digital elevation data used in this tutorial is in US survey feet, so the data will be imported in these units and the projection will be changed at a later time. . A prompt in these instructions will appear later advising the user when to make the conversion.

1. Open WMS. If WMS is already open select *File* / **New** then click **No** if asked to save changes.

2. Select *File* / **Open** to access the *Open* dialog.

3. Locate the "cequal" folder in the files for this tutorial. If needed, download the tutorial files from www.aquaveo.com.

4. Open the file named "EastCanyon.tin".

5. Right-click on "new" under "Terrain Data" and select **Display Options** to open the *Display Options* dialog.

6. On the *TIN Data* tab, deselect *Locked Vertices*, *Unlocked Vertices*, and *Triangles*.

7. Select **OK**.

8. Select *Display* / **Display Projection...**

9. In the *Display Projection* dialog, select *No projection* and "U.S. Survey Feet" for both *Horizontal* and *Vertical units*.

10. Select **OK**.

4   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** to open the *Display Options* dialog.

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. The *elevation (elev) Contour Options* dialog will open. 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", and then click on the **Populate Values** button.

9. Click **OK** in the *Value Population Method* dialog.

10. 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 **Error! Reference source not found.**.
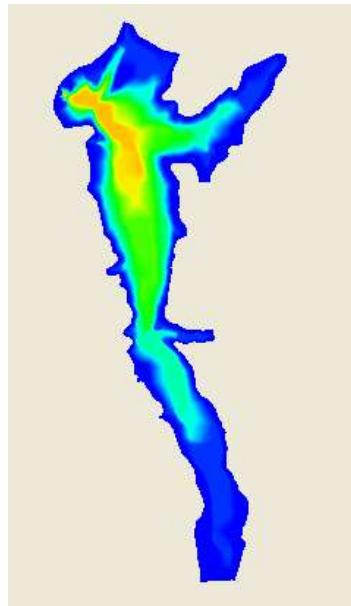
Figure 1   Reservoir Extent of East Canyon

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

4.1   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**.

4.2   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 the polygon slightly larger than the boundary generated by the maximum water surface elevation. This way, users ensure that the entire reservoir will be modeled. If users 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 "Drainage" coverage and select **Build Polygon**.

5. If prompted, click **OK** to use all arcs.

6. Select the **Select Feature Polygon** 🔲 tool.

7. Click somewhere within the reservoir. If all of the arcs have been connected correctly, the entire area will become highlighted.

5   Change Contour Display Options

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

1. Select the **Contour Options** 🔷 icon. The *elevation (elev) Contour Options* dialog will open.

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.

6   Create Branch Coverage and Branches

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

6.1   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" in the *Properties* dialog.

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**.

6.2   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 **Error! Reference source not found.** as a guide to create two feature arcs (branches) where indicated using the following two steps.

Figure 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 **Error! Reference source not found.** 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 users identify these problems. Users can also re-build the polygons after they make sure all arcs are correctly snapped. If problem persists, delete the created arcs, re-create arcs and re-build polygons.

7   Create Segment Coverage and Segments

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

7.1   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" in the *Properties* dialog.

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**.

7.2   Create Segments

Users 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 **Error! Reference source not found.** as a guide to create feature arcs (segments) where indicated using the following two steps (Remember that it is unnecessary to create segments where branches were defined previously).

Figure 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 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 the project as "EastCanyon" in the *Save As* dialog.

8   System Modeling

After the segments and the branches have been created, it is a good idea to create a conceptual model of the system. **Error! Reference source not found.** 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 the 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.
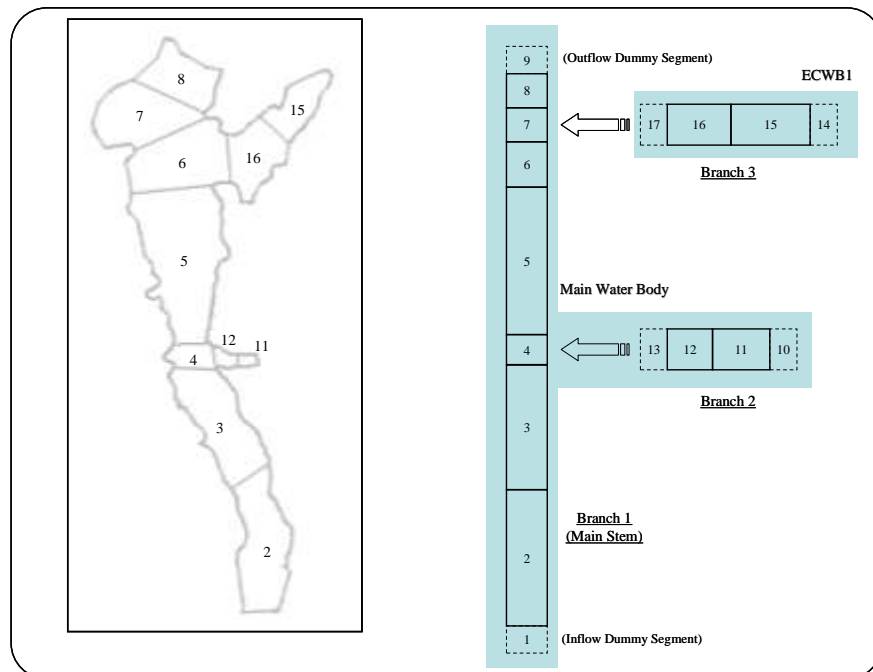
Figure 4　　System Model

8.1　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 of the *Reproject Current* dialog, change the *Horizontal* and *Vertical units* to "Meters" (make sure *No projection* is selected)..

　　4.　Select **OK**.

8.2　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* dialog.

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 **Error! Reference source not found.**.

Users 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, users can add as many points along the line as desired.

As users 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 **Error! Reference source not found.**.

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.

8.3   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 users 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.

8.4   Editing Segment Properties

Within each segment, many individual properties must be defined. These include the segment length, the layer properties, and the width computations. **Error! Reference source not found.** shows the different parameters required for the segment.
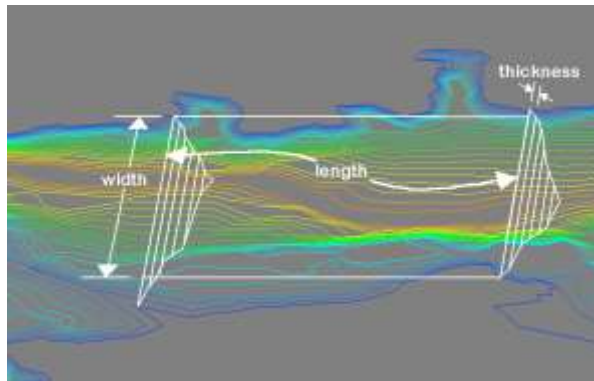
Figure 5   Segment bathymetry

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. The *CE-QUAL-W2 Layer Editor* will appear. Enter "1768" (meters) as the top elevation of the reservoir in the field labeled *Top elevation*.

NOTE: Users 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 6   CE-QUAL-W2 Layer Editor

6. Select "Layer Heights" from the *Layer Generation* drop-down menu .

7. Enter "10.0" in the *Value* field.
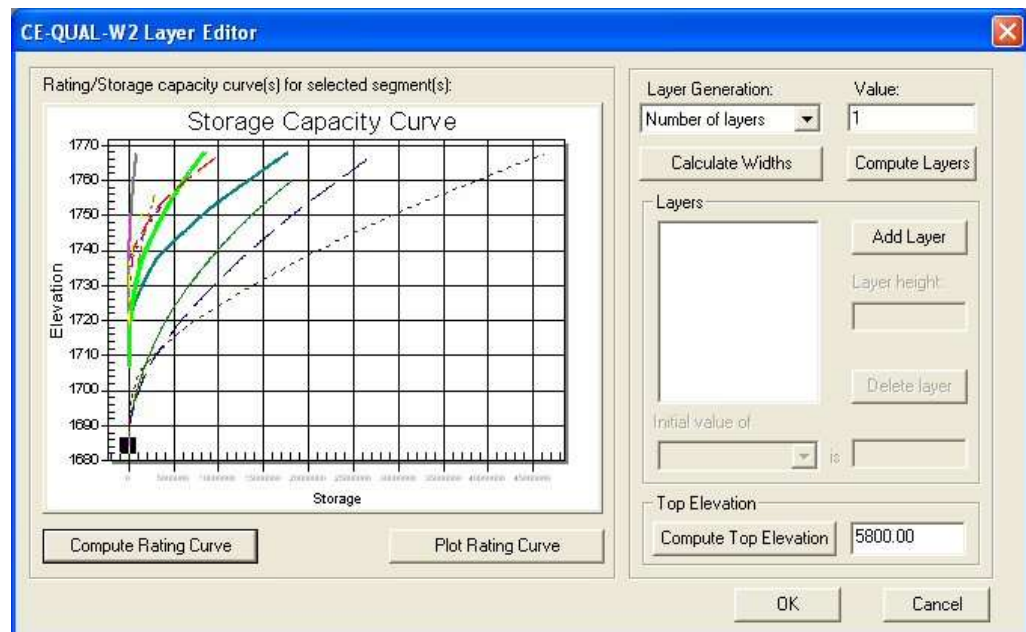
Notice that users can also enter in a number of layers instead of a layer height. If the number of layers option is chosen, then the layer height will be calculated using the following equation:

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.

9   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, Users will learn to generate/modify control file parameters.

9.1   Time Step Job Control

1. Select *CE-QUAL-W2* / **Time Step Job Control**.

2. In the *Time Step* dialog, 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 *day* 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 users have set the time step parameters along with some other general model parameters, users are ready for more model-specific parameters.

9.2   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**. The *Water Body Job Control* dialog will appear.

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** in the *Dates - - Screen Print* dialog.

9. Click on the added date and click **Edit**.

10. Change the *year* to "1998", the *month* to "5", and the *day* to "1" in the *Reference Time* dialog.

11. Click **OK**.

12. Change *Frequency* to "10".

13. Click **OK**.

14. Do the same (steps 7-13) for *Profile Plot Output* and *Snapshot Output*.

15. Click **OK** to close the *Water Body Job Control* dialog

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

9.3   Editing Segments

In this section users 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).

Users 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 users used the measure tool to measure the branches in section 8.2. If segment length and orientation are not already defined, users 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 the model in the graphics window.

9.4   Structures

In this section users will learn to add/modify structures to the model. WMS will let users add one structure for each node in the branch coverage. However, if users have multiple structures in the same place, they 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** [icon] tool, as shown by the arrow in **Error! Reference source not found.**.

Figure 7   Insert Node

3. Click on the **Select Feature Point/Node** [icon] tool.

4. Double-click on the new node to open the *Node Attributes* dialog.

5. Toggle on the *Structure* option. 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" from the drop-down list.

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 users have defined the spillway at the specified location. Users can also define other structures as needed.

9.5   Constituent Control

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

1. Select *CE-QUAL-W2 /***Water Body Job Control**. The *Water Body Job Control* will reappear.

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 *Constituent Name*.

7. Click the **Add Constituent** button.

8. Similarly, add another "Algal group" (*Constituent Name*: "Algae 2,") and two "Epiphyte groups" (*Constituent Names*: 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 users back to the *Water Body Job Control*; *Calculations* tab. Do not close this dialog; users 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, users will learn how to modify the constituent-specific kinetic coefficients.

9.6   Kinetic Coefficients

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

1. Make sure the *Calculations* tab in the *Water Body Job Control* dialog is open.

2. Click on the **Kinetic Coefficients** button to open the *Kinetic Coefficient* dialog.

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. Change *AT2* to "37.5".

6. Select "Algae 1" from the *Algal Group* box. Change *AT2* to "22.5".

Users 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, users 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.

10   Saving CE-QUAL-W2 Input Files

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

WMS generates one control file for the model. However, it can create one or multiple bathymetry files depending on how many water bodies users have in the model. In other words, users 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 from the *Save CE-QUAL-W2 Simulation* dialog.

3. For the *Control filename*, make sure that the prefix is no longer than 8 characters long.

NOTE: users 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 users might have in the 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.

11   Conclusion

In this exercise users were acquainted with the CE-QUAL-W2 interface in WMS. This included 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