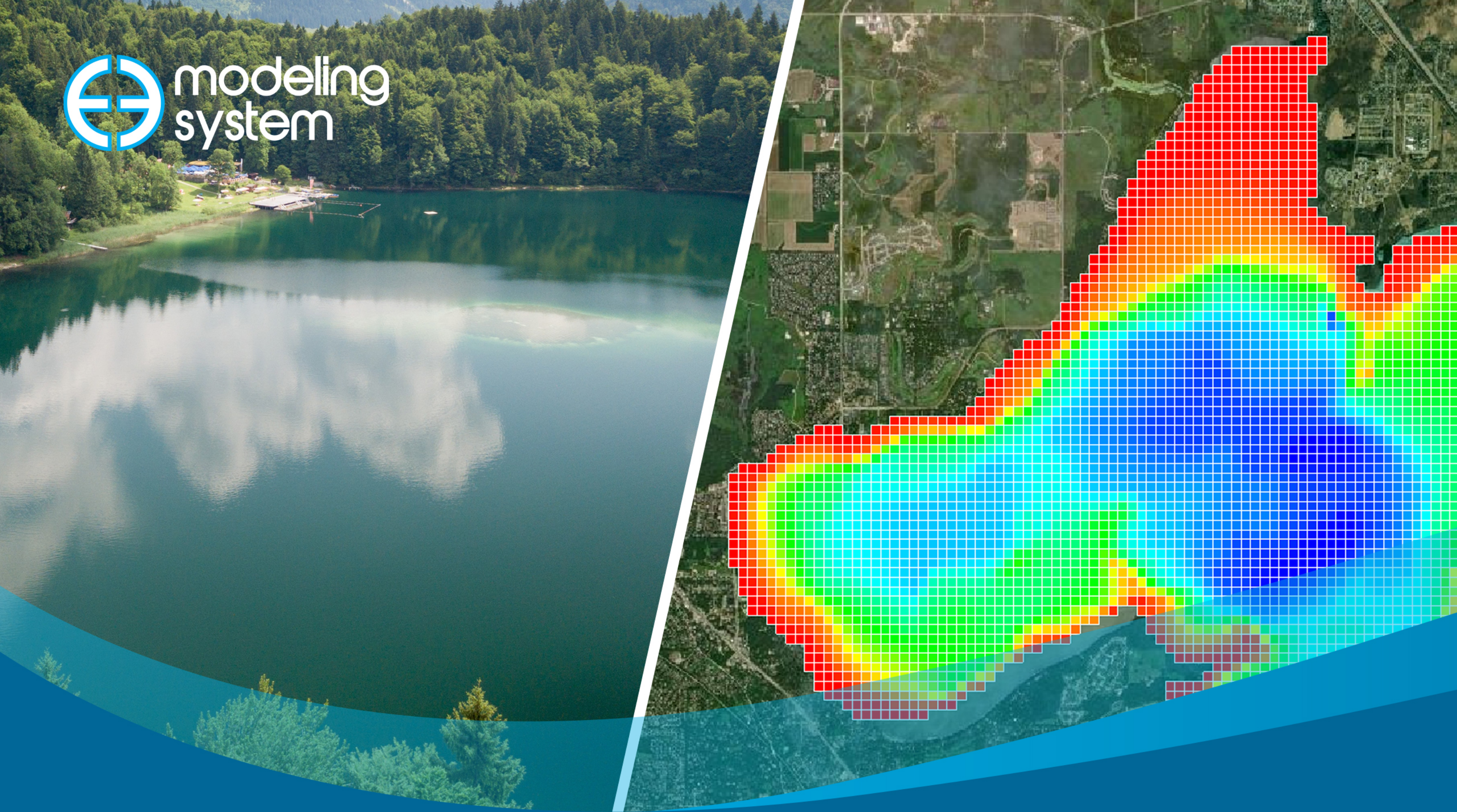

Introduction to the EEMS BASINS Plugin

May 8th 2023

www.dsi.llc

Contents

List of Figures ii

List of Tables iv

List of Abbreviations v

1 Installing the EEMS BASINS Plugin 1

2 Accessing the EEMS BASINS Plugin 3

3 Building a BASINS Project 4
- 3.1 Create a New BASINS Project 4
- 3.2 Download Data 6

4 Building an EEMS Model 13
- 4.1 Generate Grid and Import Time-series 13

5 Build and Run Model with EE 20
- 5.1 Open Model in EFDC+ Explorer (EE) 20
- 5.2 Assign Model Bathymetry 21
- 5.3 Assign Water Elevation 23
- 5.4 External Forcing Data 26
- 5.5 Hydrodynamic Settings 41
- 5.6 Model Timing 45
- 5.7 Run Model 47
- 5.8 View Model Output 48

List of Figures

1.1 EEMS BASINS plugin setup wizard 1
1.2 Completing the EEMS BASINS plugin setup 2

2.1 Accessing the EEMS plugin 3

3.1 Welcome to BASINS 4
3.2 Build new BASINS project 5
3.3 Projection properties 5
3.4 Create a new project 6
3.5 Download data 7
3.6 Export Lake Mendota shapefile 8
3.7 Zoom to Lake Mendota shapefile 8
3.8 Selected NWIS discharge stations near Lake Mendota. 9
3.9 Download daily discharge data for the NWIS Stations. 10
3.10 Select NLDAS Grid 11
3.11 Download NLDAS Data 12

4.1 EFDC+ Model form 14
4.2 Import flow wdm file 15
4.3 The pop-up message of unit conversion 15
4.4 Assign flow series to the Lake Mendota model 16
4.5 Approximate Locations of flow boundaries 17
4.6 Folder with the EFDC+ Model of Lake Mendota. 18
4.7 Export grid 19

5.1 Open Lake Mendota model 21
5.2 Assign bathymetry(1) 22
5.3 Assign bathymetry(2) 23
5.4 Assign bathymetry(3) 23
5.5 Assign water surface elevation 24
5.6 Apply Cell Properties: Water Surface Elevations 25
5.7 Report of number of assigned cells for initial water surface elevation 25
5.8 Assign Outflow 26
5.9 Add New Data Series 27
5.10 Import ungaged flow 28
5.11 Parameters of ungaged flow 29
5.12 Import wind boundary time series (1) 29

5.13 Import wind boundary time series (2) 30
5.14 Parameters of Dane County Airport 31
5.15 Add Atmospheric Data Series (1) 32
5.16 Add Atmospheric Data Series (2) 33
5.17 Parameters of Dane County Airport meteorological station 34
5.18 Add Groundwater Data Series(1) 35
5.19 Add Ground Water Data Series(2) 36
5.20 Groundwater summary 37
5.21 Groundwater interaction 38
5.22 Apply Cell Properties Groundwater 39
5.23 Add new boundary group 40
5.24 Assign surface balance boundary group 41
5.25 Turbulent Diffusion 42
5.26 Turbulent Intensity 43
5.27 Applying Bottom Roughness 44
5.28 Hydrodynamic - Shallow water setting 45
5.29 Simulation timing options 46
5.30 EFDC+ Explorer Linkage 47
5.31 EFDC+ Run 48
5.32 Online Background map for Lake Mendota 49
5.33 Add velocity vector layer in 2DH View 50

List of Tables

4.1 Assign flow time series to boundary groups 17
5.1 Parameters of ungaged flow station 28

List of Abbreviations

BASINS	Better Assessment Science Integrating Point And Nonpoint Sources
cfs	Cubic Foot Per Second
cms	Cubic Meter Per Second
EE	EFDC+ Explorer
EEMS	EFDC+ Explorer Modeling System
EFDC+	Environmental Fluid Dynamics Code Plus
HUC	Hydrologic Unit Code
NLDAS	North American Land Data Assimilation System
NWIS	National Water Information System
RMC	Right Mouse Click
WDM	Watershed Data Management

Introduction EEMS BASINS Plugin

Better Assessment Science Integrating point and Nonpoint Sources (BASINS) is a multipurpose environmental analysis system and has been made freely available by its developer, the US EPA. It is widely used as it provides a convenient framework for integrating modeling tools and environmental spatial and tabular data into a geographic information system (GIS) interface. From the BASINS interface, you can automatically download vital data to facilitate the setup of new projects. The tool accesses data such as Digital Elevation Models (DEMs), hydrography, land use, meteorological data, monitoring data, and soil maps from EPA, USGS, and many other cloud-based locations.

DSI has recently developed a plugin for BASINS that allows users to generate the grid and build an Environmental Fluid Dynamics Code Plus (EFDC+) model directly from within the BASINS interface. Using a shape file of your waterbody displayed in the BASINS interface, you can call the plugin and build a grid to your required resolution. This tool allows users to extract the essential data from Watershed Data Management (WDM) format and create the input files for EFDC+. After this, users can specify the boundary locations and assign the flow time series, and the plugin will convert them into the correct format for EFDC+. Meteorological data such as wind speed, direction, rainfall, and solar radiation, cloud cover can also be exported from BASINS into the file format needed by EEMS.

The EEMS plugin for BASINS provides an important link between watershed models and receiving water hydrodynamic models. Users will be able to go from extensive basin-scale studies to more in-depth analyses of particular water bodies and the impact of pollutant loading. This plugin is the first step in what we envision to be greater integration between hydrologic and hydrodynamic models, facilitating improved data exchange and ease of use for those seeking solutions to complex environmental questions.

The plugin is now available for download from the EEMS website https://www.eemodelingsystem.com/user-center/downloads.

What follows is a detailed description of how to install the EEMS plugin and a demonstration of using this tool to build a hydrodynamic model of Lake Mendota. Similar steps will be required for any model.

Chapter 1

Installing the EEMS BASINS Plugin

After downloading the EFDC+ Explorer Modeling System (EEMS) Better Assessment Science Integrating Point and Nonpoint Sources (BASINS) Plugin installation package, double-click on it, and the EEMS BASINS Plugin Setup Wizard will appear (Figure 1.1).

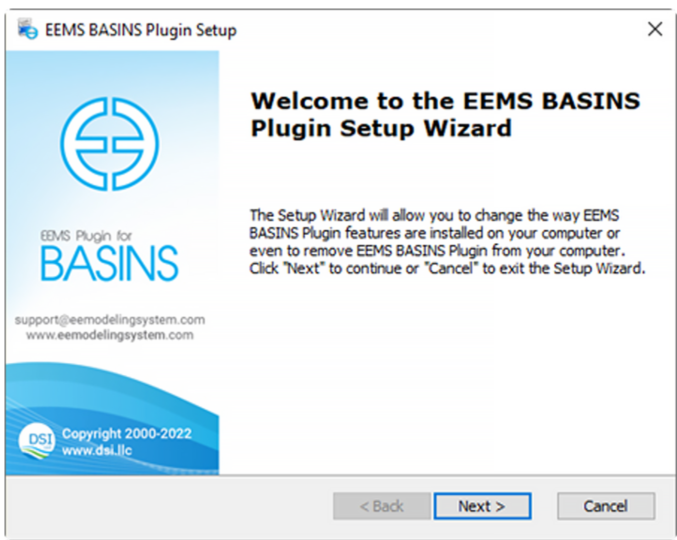

Figure 1.1: EEMS BASINS plugin setup wizard

Click the *Next* button, and follow the steps to install the plugin. Upon completion of the Installation Wizard, you will be prompted with the message "Completing the EEMS BASINS Plugin Setup Wizard" (Figure 1.2). Click the *Finish* button to finish the installation.

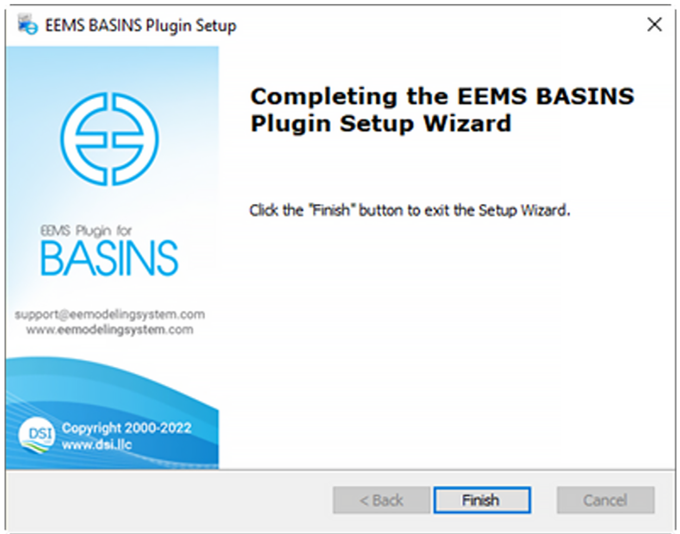

Figure 1.2: Completing the EEMS BASINS plugin setup

Chapter 2

Accessing the EEMS BASINS Plugin

Open BASINS from the Windows Start Menu, click on *Plug-ins* in the main menu, and turn on the EEMS plugin (Figure 2.1). The EEMS plugin will then be available in the BASINS toolbar.

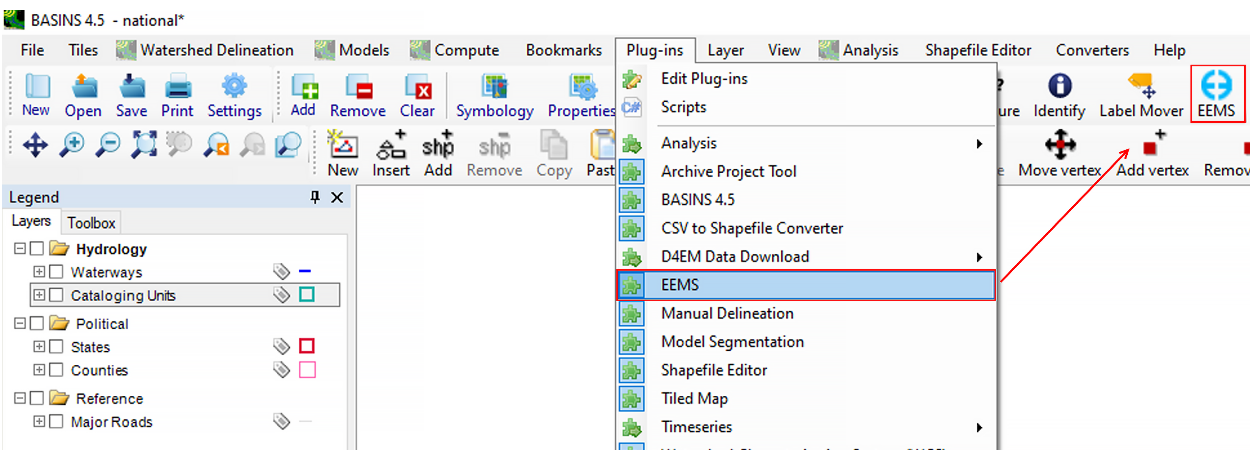

Figure 2.1: Accessing the EEMS plugin

Chapter 3

Building a BASINS Project

The section is a demonstration of how to build a BASINS project for Lake Mendota and download the necessary data from BASINS in order to set up a hydrodynamic model in EEMS.

3.1. Create a New BASINS Project

From the *Start* menu under *Programs*, select BASINS. The *Welcome to BASINS* window will open (Figure 3.1). Click on *Build New Project*. **Note:** Do not close the *Build New BASINS 4.5 Project* window popped up for the following steps below.

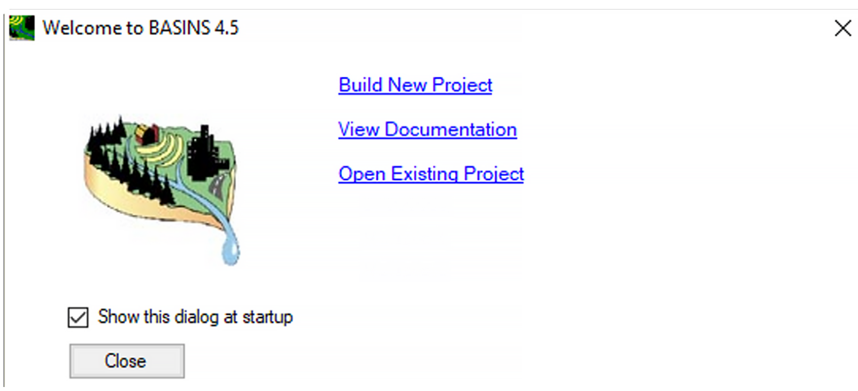

Figure 3.1: Welcome to BASINS

- Zoom in and click the Hydrologic Unit Code (HUC) 07090002 (Near the Southern border of Wisconsin) unit from the map (Figure 3.2).
- Click the *Build* button in the *Build New BASINS 4.5 Project* window and save the project.

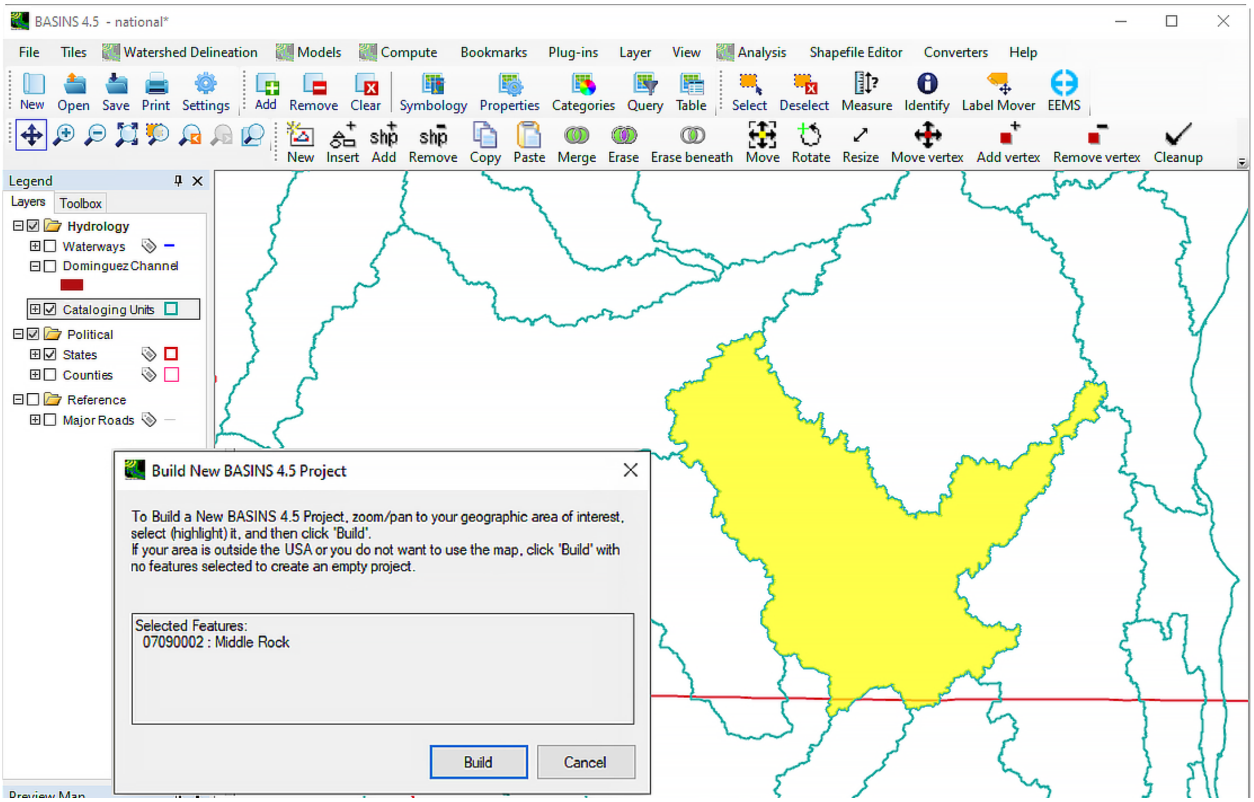

Figure 3.2: Build new BASINS project

Set the *Projection Properties* as shown in Figure 3.3, and *click OK* to close the form. If you have used BASINS before, it may have this form set up according to the previous project, so be sure to double-check this form. Once you click *OK*, BASINS will download initial data for the selected HUC, and create a project.This may take up to 5 minutes.

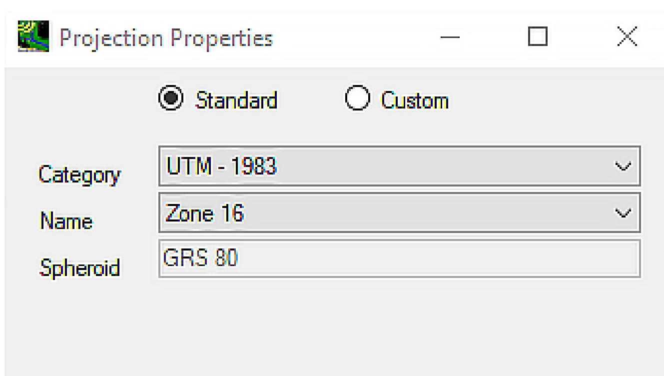

Figure 3.3: Projection properties

Following the initial data download, extraction, and projection, a dialog box will pop up to confirm the creation of the project (Figure 3.4). The contents of the dialog box may differ based on the file and folder names specified above. Click OK and the new BASINS project will open.

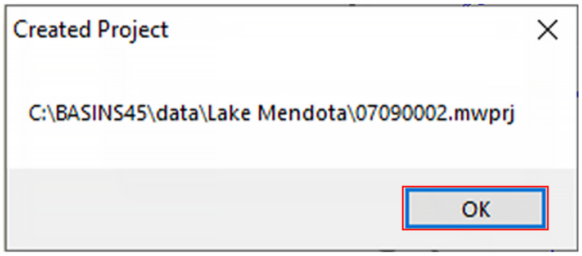

Figure 3.4: Create a new project

3.2. Download Data

BASINS supports downloading data from several online repositories from the interface. This section shows how to use *Data Download* form to retrieve flow and meteorological data and create a meteorological Watershed Data Management (WDM) file.

1. From the *File* menu in BASINS, select *Download Data*. The *Download Data* form pops up (Figure 3.5).
2. In the *Download Data* form, set the *Region to Download* option as *View Rectangle*, and check the box *Hydrography* in *National Hydrography Dataset Plus v2.1*, which is the most recent hydrology dataset available.
3. Select *Discharge* to download the *Station Locations*.
4. Check the box *Grid* to download *North American Land Data Assimilation System*.
5. Click the *Download* button to start downloading.

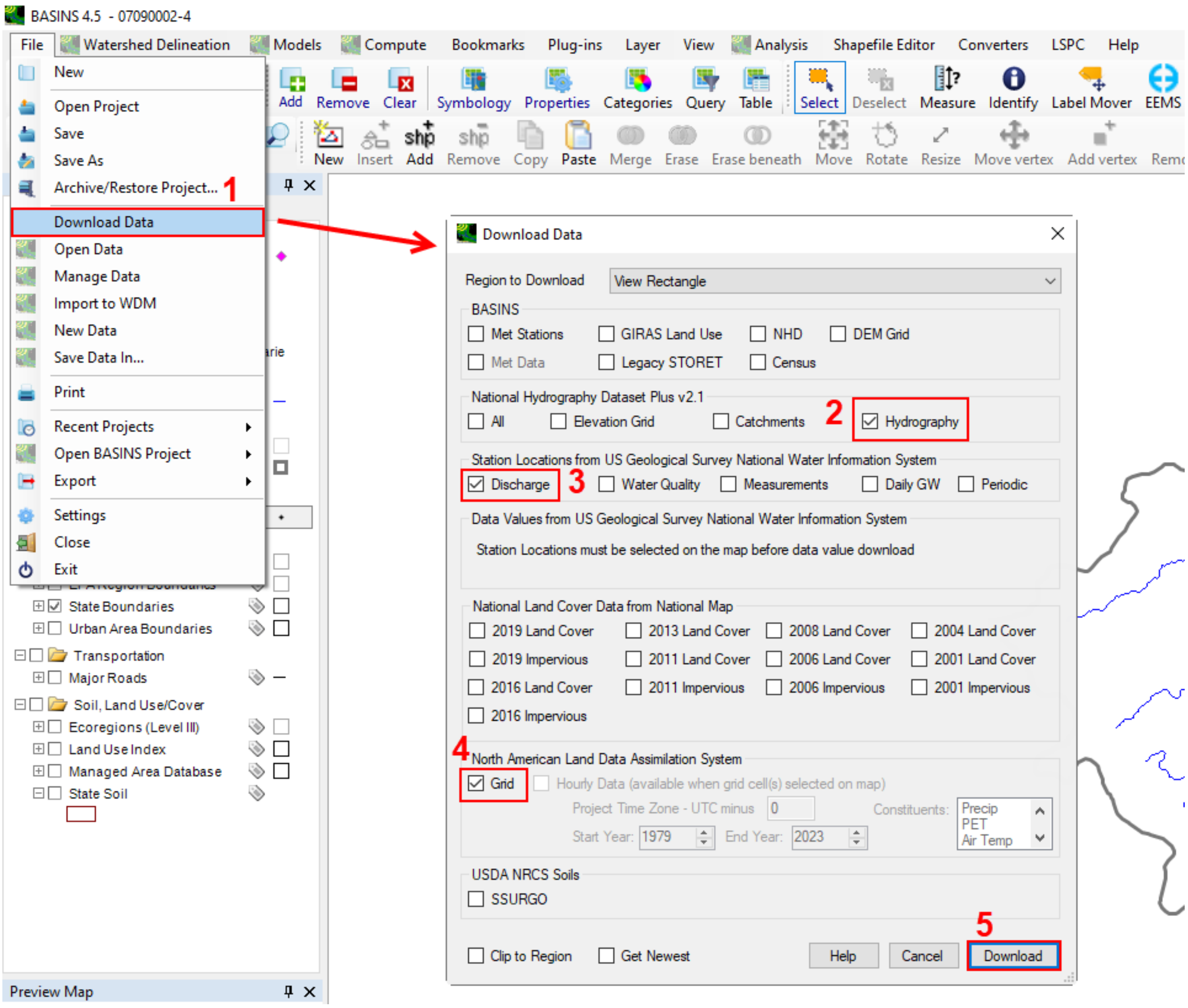

Figure 3.5: Download data

After downloading, BASINS displays the waterbody features in the catchment. You may have to select the options on the left panel to display the datasets of interest. The following steps illustrate how to export the Lake Mendota shapefile (Figure 3.6).

1. Right mouse click the *Waterbody Features* layer in the Legend and select *Attribute Table Editor*.
2. In the *Attribute Table Editor* tab, find the element that has a *GNIS Name* as *Lake Mendota*. The selected row is highlighted. **Note:** In case BASINS selection tool doesn't work, you may choose to select Lake Mendota using "Query" option under *Selection* option in the Attribute Table Editor. The query should be *[GNIS_ID] = "1792750"*. Do not sort by any attributes.
3. Click *Selection* in the toolbar and select *Export Selected Features*.
4. Save the selected waterbody features as a shapefile named **Lake Mendota.shp** in the default folder.

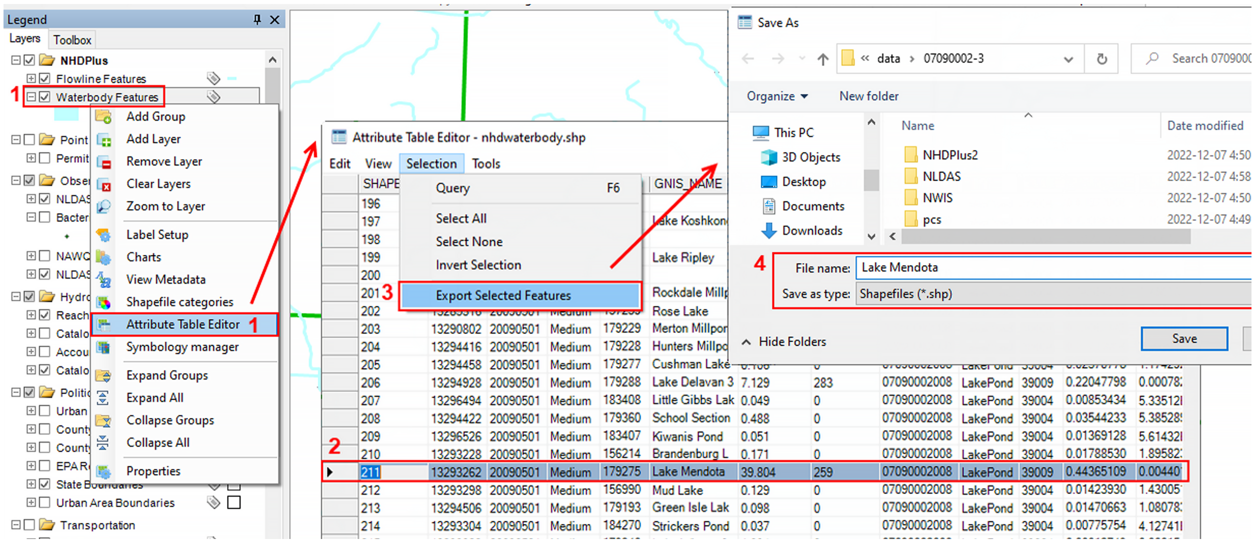

Figure 3.6: Export Lake Mendota shapefile

After exporting the shapefile, you can zoom to the shapefile by RMC on the layer can click *Zoom to Layer* as shown in Figure 3.7

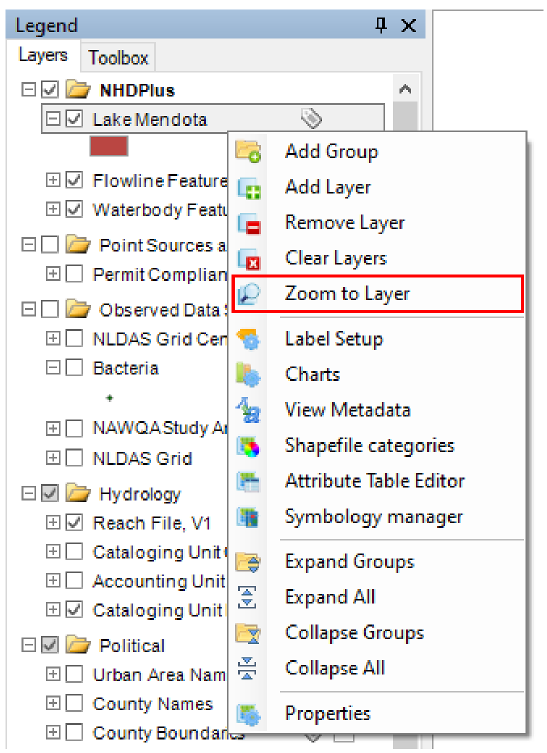

Figure 3.7: Zoom to Lake Mendota shapefile

Next steps, you can download additional data from the BASINS interface as described below (Figure 3.8).

1. Select the *National Water Information System (NWIS) Daily Discharge Stations* layer in the left panel. The selected layer will then be highlighted.
2. Click the *Select* button in the main toolbar to change to selection mode. Then select all the discharge stations in the region near Lake Mendota, and hold the *Ctrl* key to select multiple stations. The selected stations are highlighted in a different color.

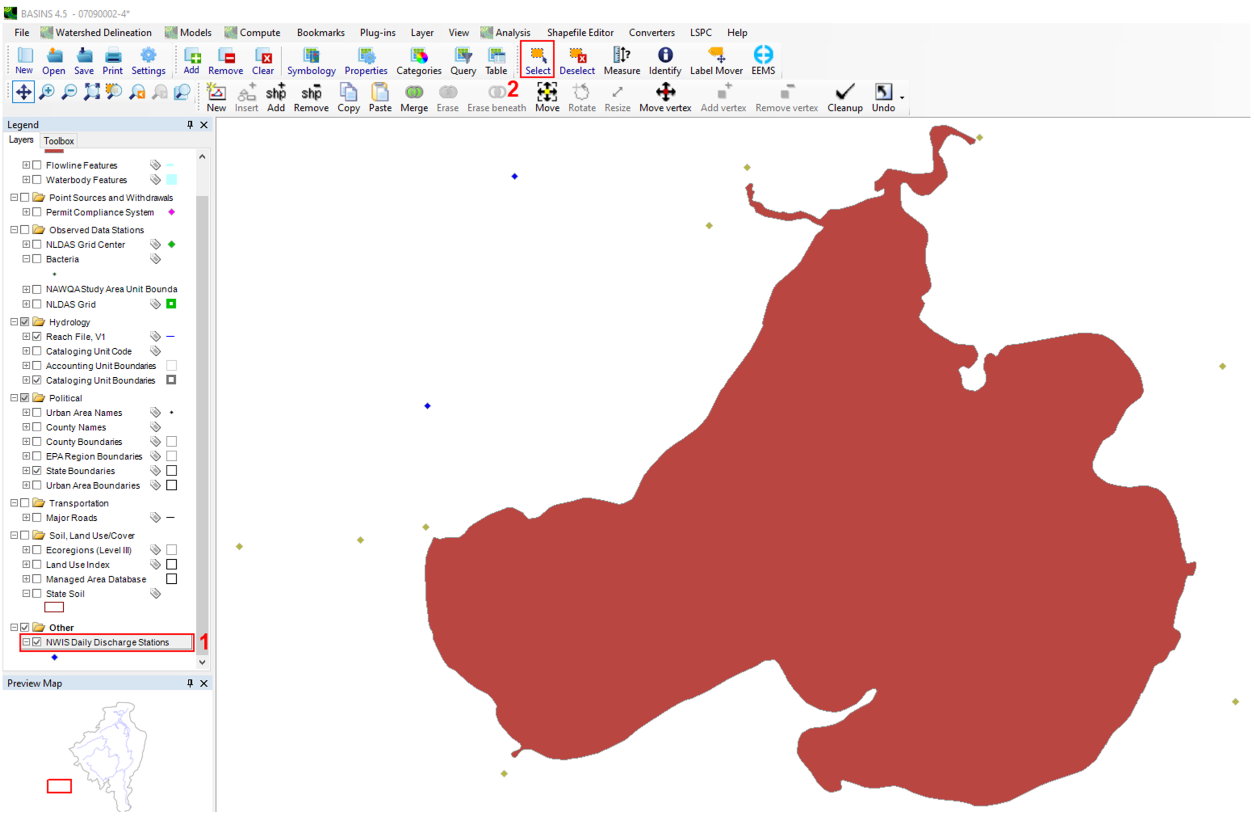

Figure 3.8: Selected NWIS discharge stations near Lake Mendota.

When the station locations on the map are selected, the data values from US Geological Survey National Water Information System will be available. The following steps (Figure 3.9) show how to download daily discharge in the selected stations.

1. From *File*, select the *Download Data* option.
2. As the NWIS stations are selected, the *Data Values from US Geological Survey National Water Information System* is now available; check the box for *Daily Discharge*.
3. Click the *Download* button, and the *Daily Discharge Processing Options* form will pop up.
4. Select the option *Add data to new WDM file* and browse to the folder (if needed). Click *OK* to start downloading. The process may take a maximum of 5 minutes. A dialog box will pop up to confirm the downloaded data and the creation of the WDM file.

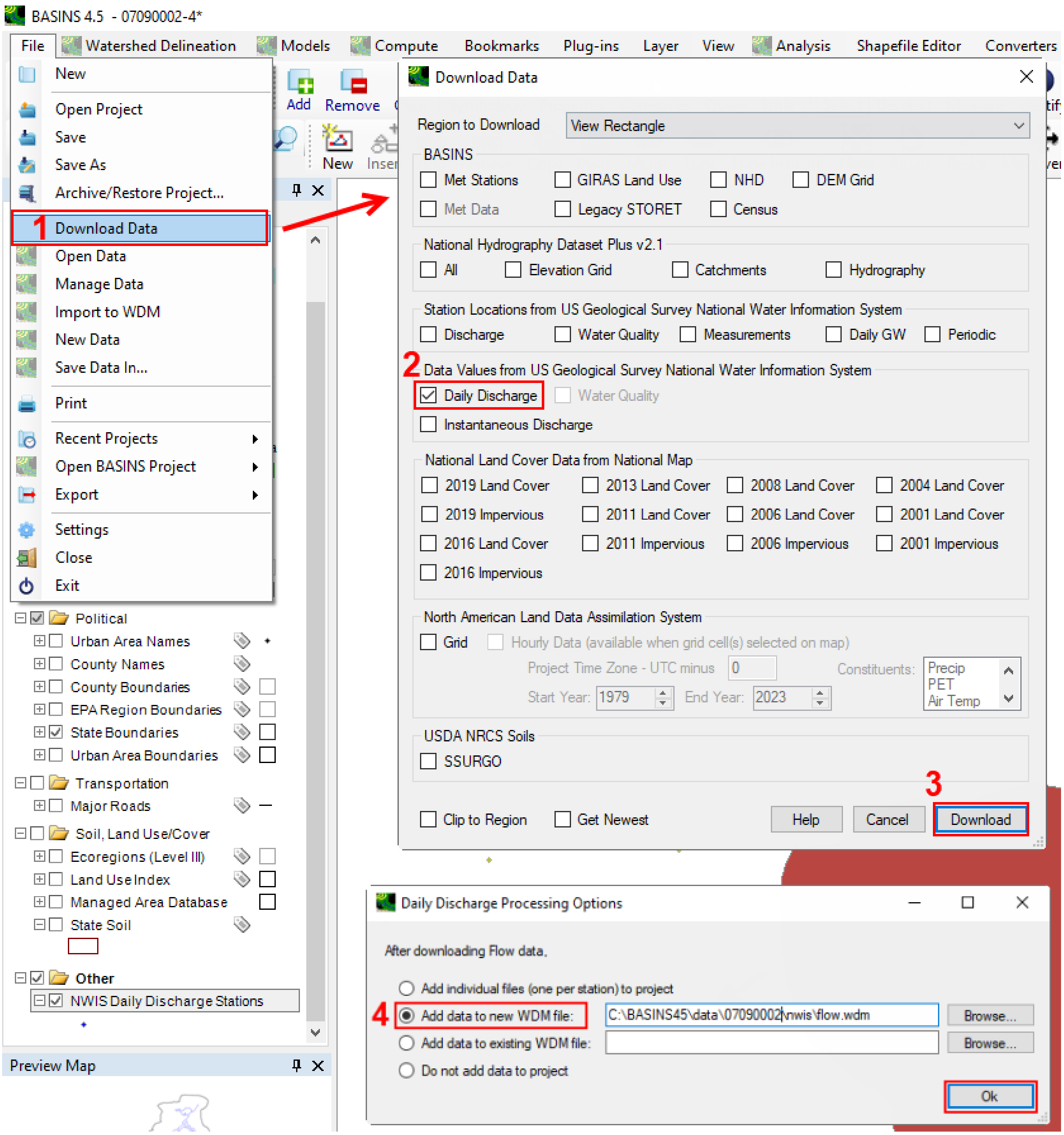

Figure 3.9: Download daily discharge data for the NWIS Stations.

In BASINS, North American Land Data Assimilation System (NLDAS) data is also available and can be directly downloaded from the interface, as described below (Figure 3.10).

1. Select the *NLDAS Grid* layer in the layer list.
2. Click *Select* button in the main toolbar. Select the grids that cover Lake Mendota, and hold the *Ctrl* key to select multiple grid cells. The selected grids are highlighted in yellow.

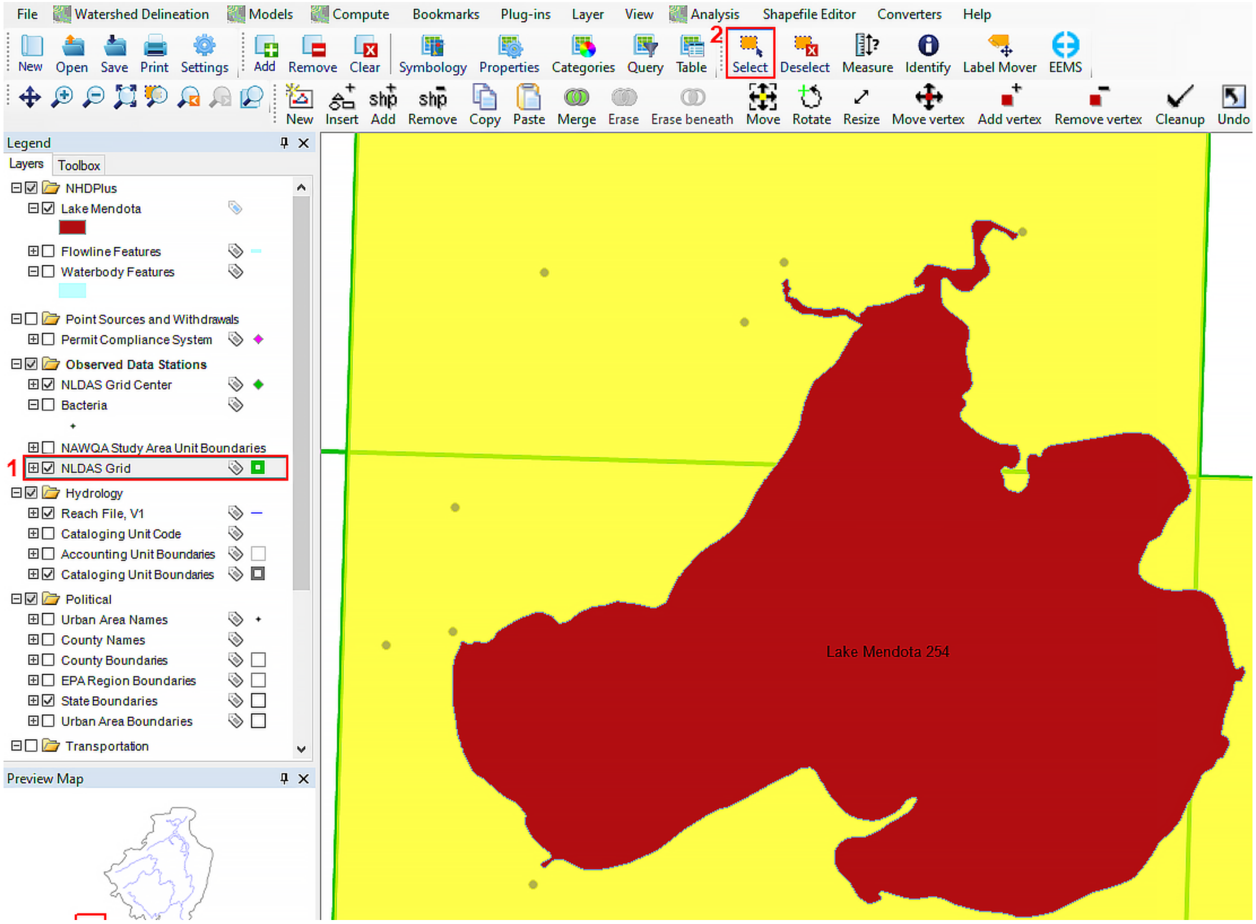

Figure 3.10: Select NLDAS Grid

The following steps show how to download available NLDAS data (Figure 3.11).

1. From the *File* menu, select *Download Data* option
2. In the *Download Data* form, check the *Hourly Data* box under *North American Land Data Assimilation System*. Since our project location is in Central Time Zone, which is 6 hours behind UTC, enter 6 in the box to correct for the time zone.
3. Click the *Download* button.
4. The *NLDAS Processing Options* window pops up. Check the option *Add data to new WDM file* and save as nldas.wdm file. Wait for the download to complete, which may take several minutes. After the download is complete, a dialog box will pop up to confirm the data download and creation of the WDM file.

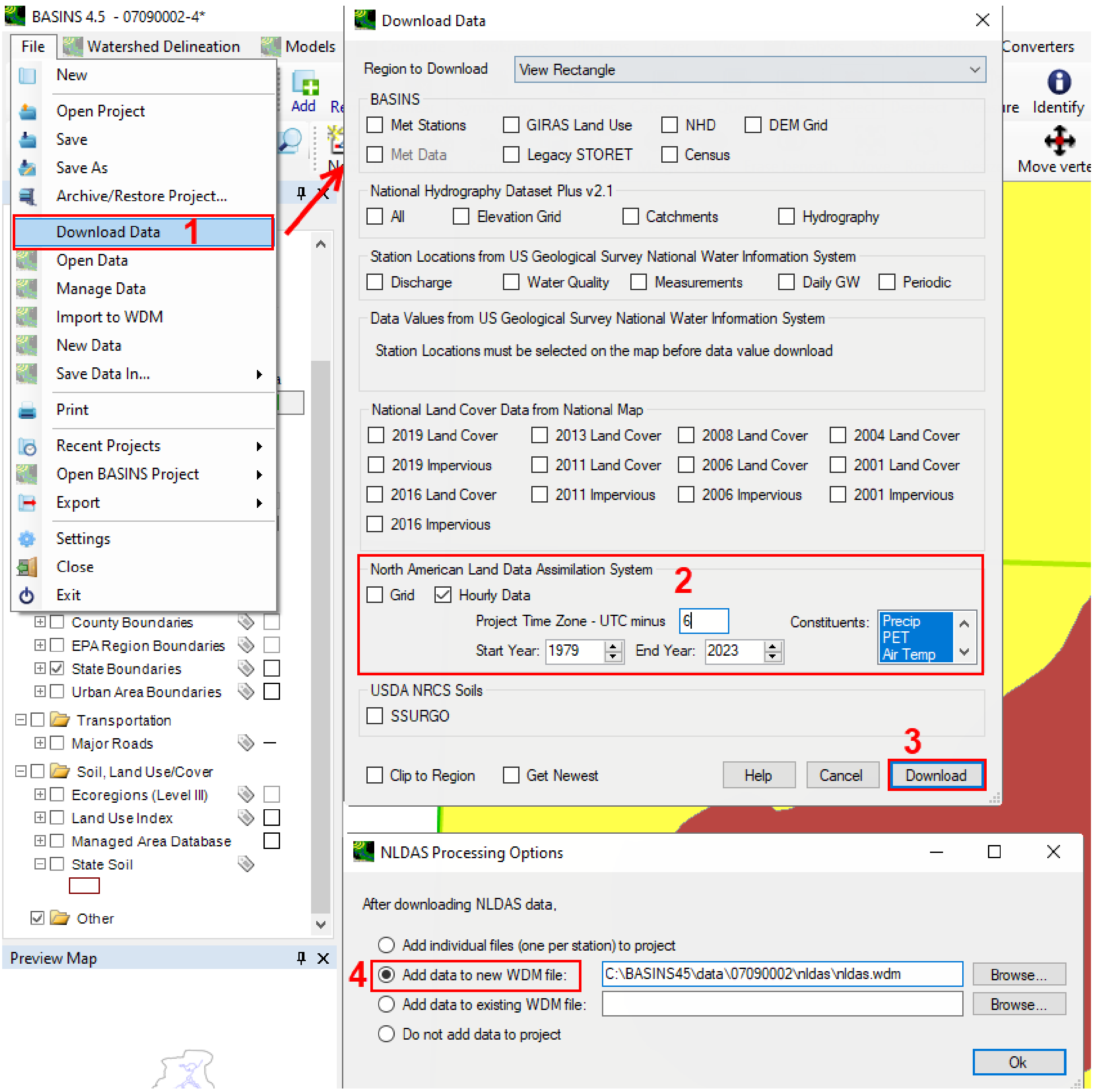

Figure 3.11: Download NLDAS Data

After downloading all data, please save the BASINS project again by clicking *Save* options in the main menu.

Chapter 4

Building an EEMS Model

The following steps describe how to use the EEMS plugin in BASINS to build a hydrodynamic model of Lake Mendota.

4.1. Generate Grid and Import Time-series

After activating the plugin, click on the EEMS button in the toolbar to open the *EFDC+ Model* form (Figure 4.1).

1. In the *Grid Options*, select *Generate Uniform Grid* to generate a Cartesian grid.
2. The drop-down *Shapefile* lists all the shapefiles available in the BASINS project. For this example, select the **LakeMendota.shp** shapefile. When the shapefile is loaded, it will be shown in the *Bounding Polygons* frame on the right side.
3. In the *Uniform Grid Options*, set the *Number of Cells* as 50 by 50. The *UTM Zone* and the *Lower-Left* and *Upper-Right* coordinates of the grid will be updated automatically based on the shapefile. **Note:** We recommend developing a low-resolution model for training and testing purposes.
4. Click the *Generate* button to build the grid and then click on *Remove Dry* to remove cells outside the polygon. The grid is displayed on the right side of the form.
5. Select a folder to save the Environmental Fluid Dynamics Code Plus (EFDC+) model by clicking on the button next to *EFDC+ Model Folder*.
6. Click *Import Timeseries* to extract WDM format data from the BASINS project.

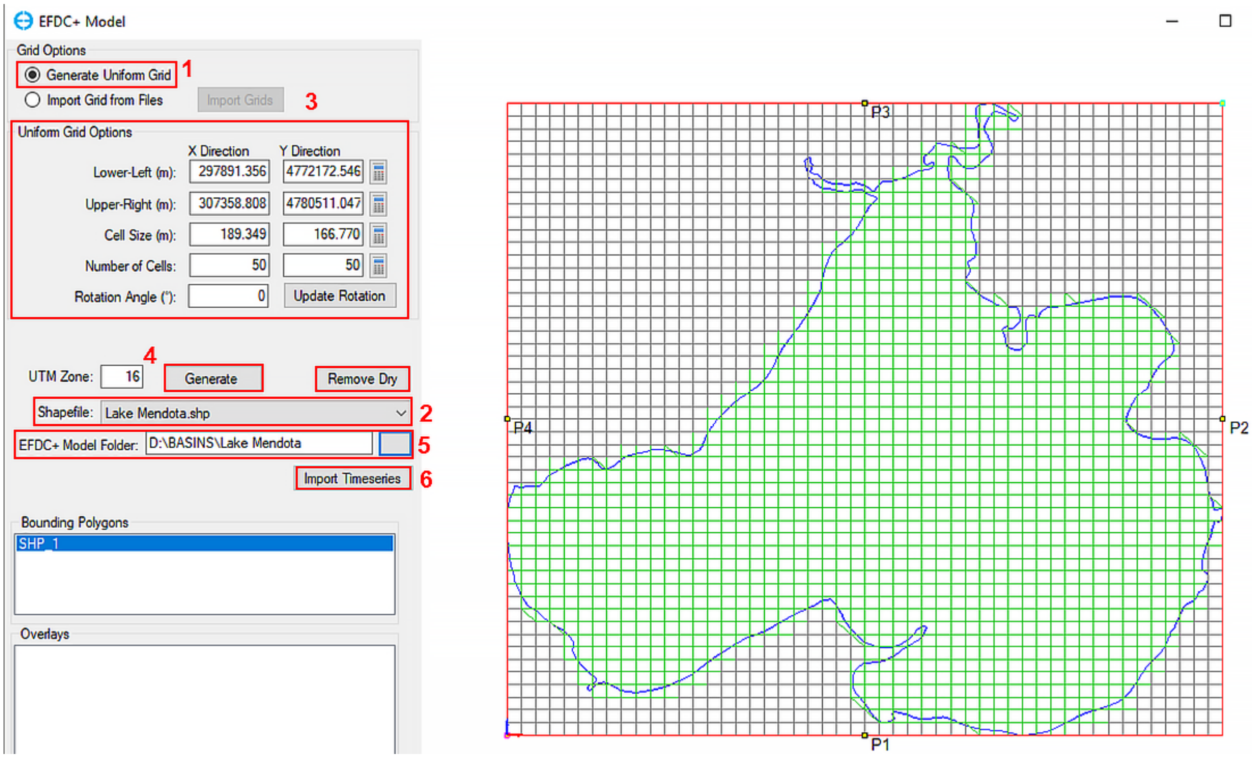

Figure 4.1: EFDC+ Model form

The *Import Time Series* form allows the user to import meteorological and flow data available in the BASINS project (Figure 4.2). However, in this exercise, we will only import flow data. To import the flow time series data:

1. Click the *Import Flow WDM File* and browse to the "flow.wdm" file in the NWIS folder under the BASINS project folder.
2. The file location, number of time series, and data are summarized in the *WDM File* frame.
3. Click *Choose Time Series*.
4. In the *Choose Flow Time Series* window, a list of time series in the flow.wdm file is shown on the left of the form. Select the data of interest and then click *OK*. For this project, we will use five flow time series as listed below. As you will see later, these flow timeseries define major inflows and outflows to the Lake Mendota.

   - OBSERVED 05427850 FLOW
   - OBSERVED 05427910 FLOW
   - OBSERVED 05427930 FLOW
   - OBSERVED 05427948 FLOW
   - OBSERVED 05428500 FLOW

   Note: If some flow data series are not displayed in the list, it means those data series were not downloaded in the previous steps. In this case, you may go back to the NWIS daily discharge download part (Figure 3.8 and Figure 3.9) then select more stations around Lake

Mendota to download the discharge data.

5. Flow data downloaded by BASINS is typically in cubic foot per second (cfs), and will be converted to cubic meter per second (cms) as required by EFDC+. If you imported data from another source and need a different conversion factor, you may have to select a different dropdown option.

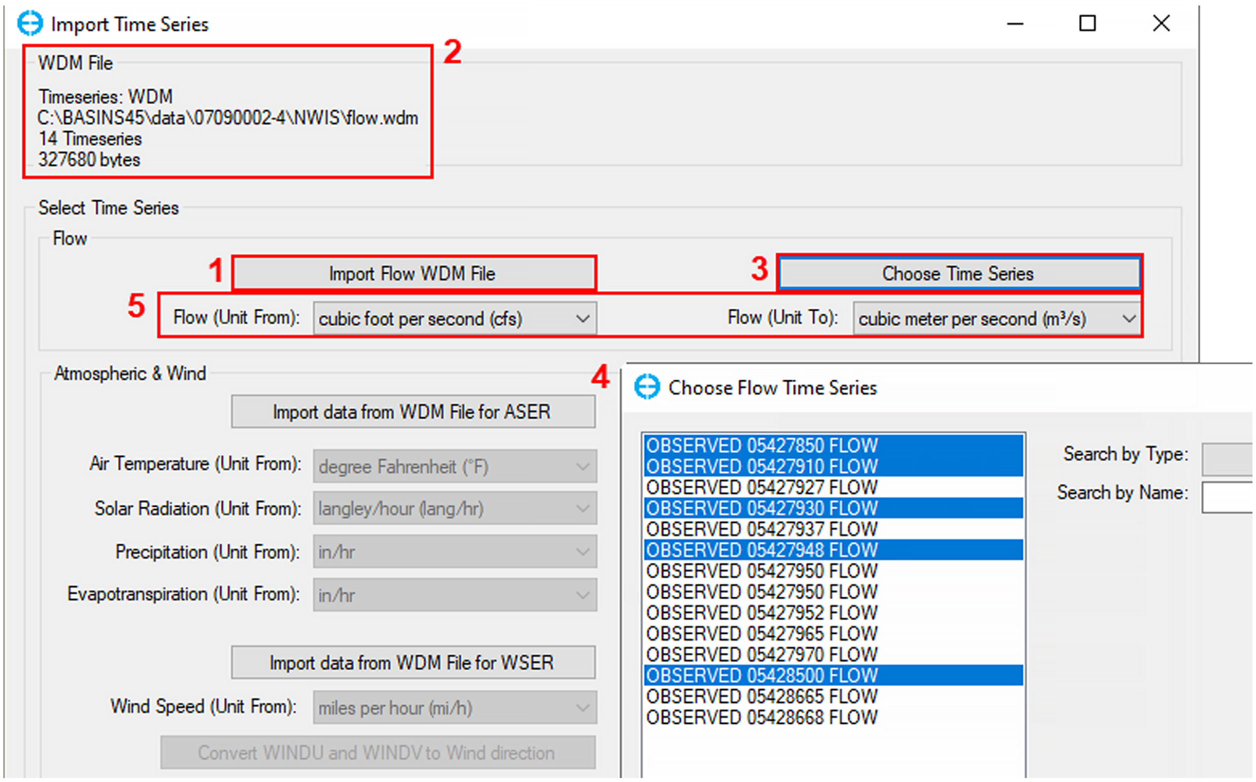

Figure 4.2: Import flow wdm file

Click *OK* to close the *Import Time Series* window, and a pop-up message is displayed to confirm that units of flow data have been converted (Figure 4.3). Click *OK* to close the message and return to the *EFDC+ Model* window.

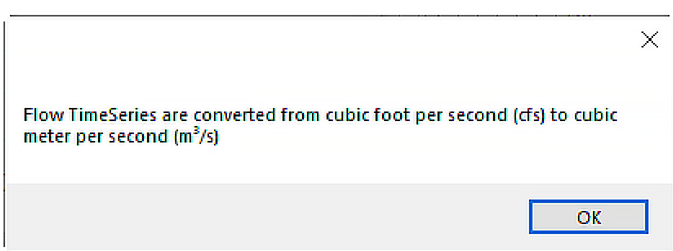

Figure 4.3: The pop-up message of unit conversion

In the following steps, you will specify the boundary locations for Lake Mendota and assign the flow time series (Figure 4.4).

1. Right Mouse Click (RMC) on one of the active cells (green color) near the north end of the Lake and select *Assign Flow Series*.
2. In the *Flow Boundary Conditions* window, set the *Group Name* as *Yahara River* .
3. Select the flow time series *OBSERVED 05427850 FLOW* from the drop-down list of the *Flow Table*.
4. Click *OK* to close the form.

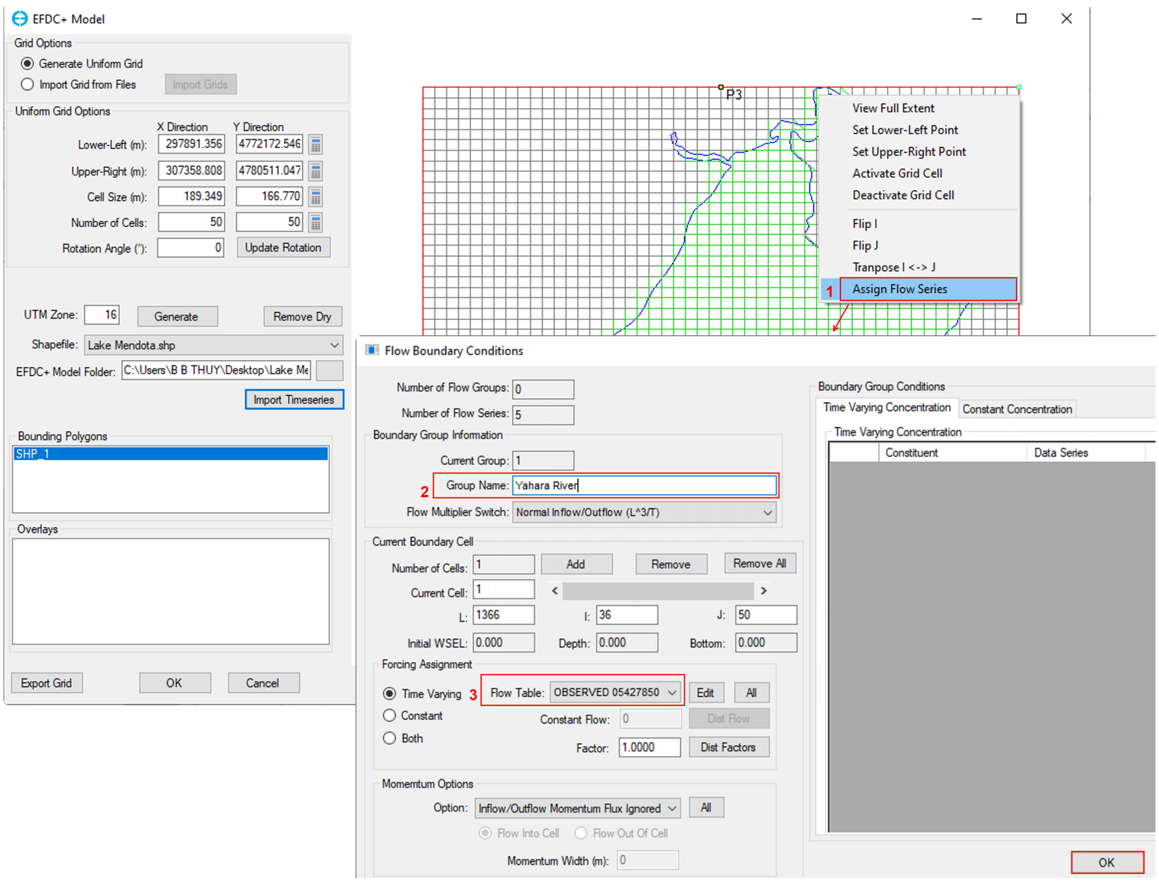

Figure 4.4: Assign flow series to the Lake Mendota model

Repeat steps from (1) to (4) to assign other flow time series to the model cells. Figure 4.5 shows the approximate locations of all the boundary inflows and the outflow, and Table 4.1 presents the boundary groups of the flow time series.

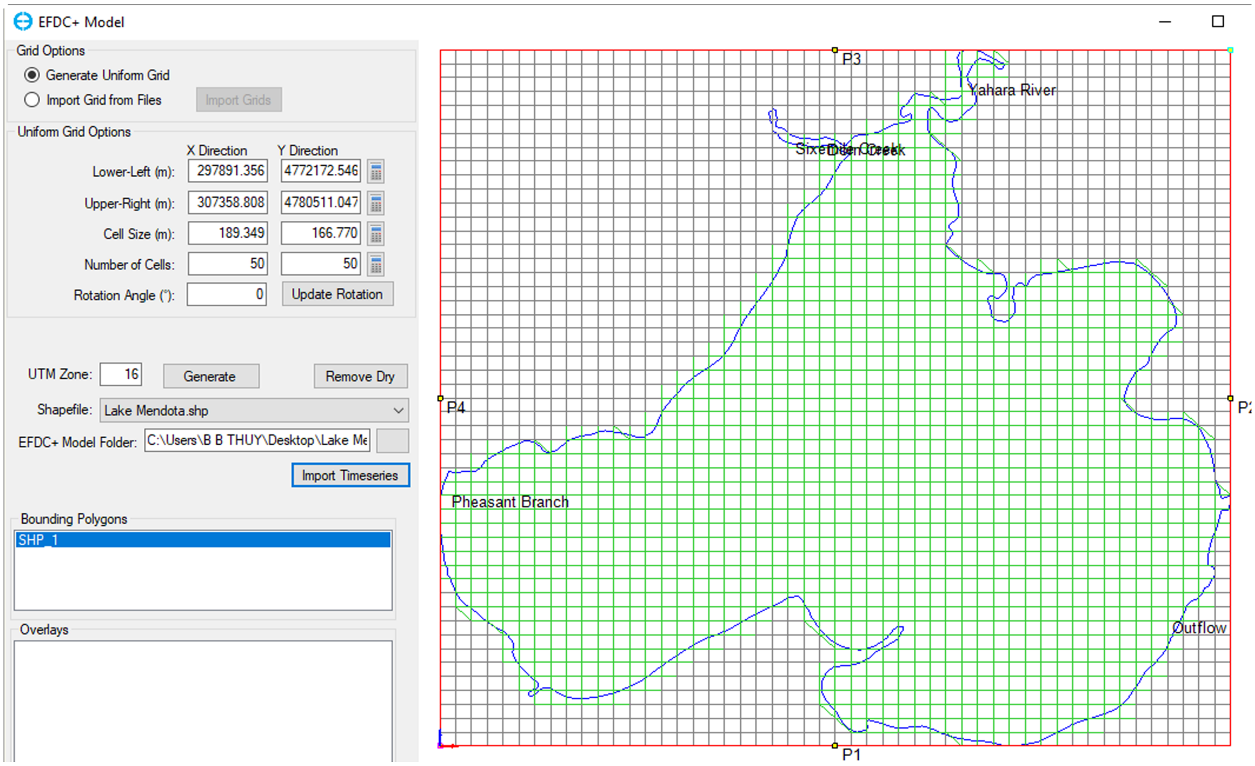

Figure 4.5: Approximate Locations of flow boundaries

Table 4.1: Assign flow time series to boundary groups

Boundary Group Name	**Flow Time Series**	**Location**
Yahara River	OBSERVED 05427850 FLOW	North End
Sixmile Creek	OBSERVED 05427910 FLOW	North West Arm
Dorn Creek	OBSERVED 05427930 FLOW	North West Arm
Pheasant Branch	OBSERVED 05427948 FLOW	West
Outflow	OBSERVED 05428500 FLOW	South East

After assigning the flow time series, click *OK* in the*EFDC+ Model* form. Then, the EFDC+ model folder will appear with the model input files generated (Figure 4.6).

Note: In some cases, the model files may not be generated if the EFDC+ model folder is located in Desktop. In this case, you may re-specify the model folder location by clicking on the button next to *EFDC+ Model Folder*.

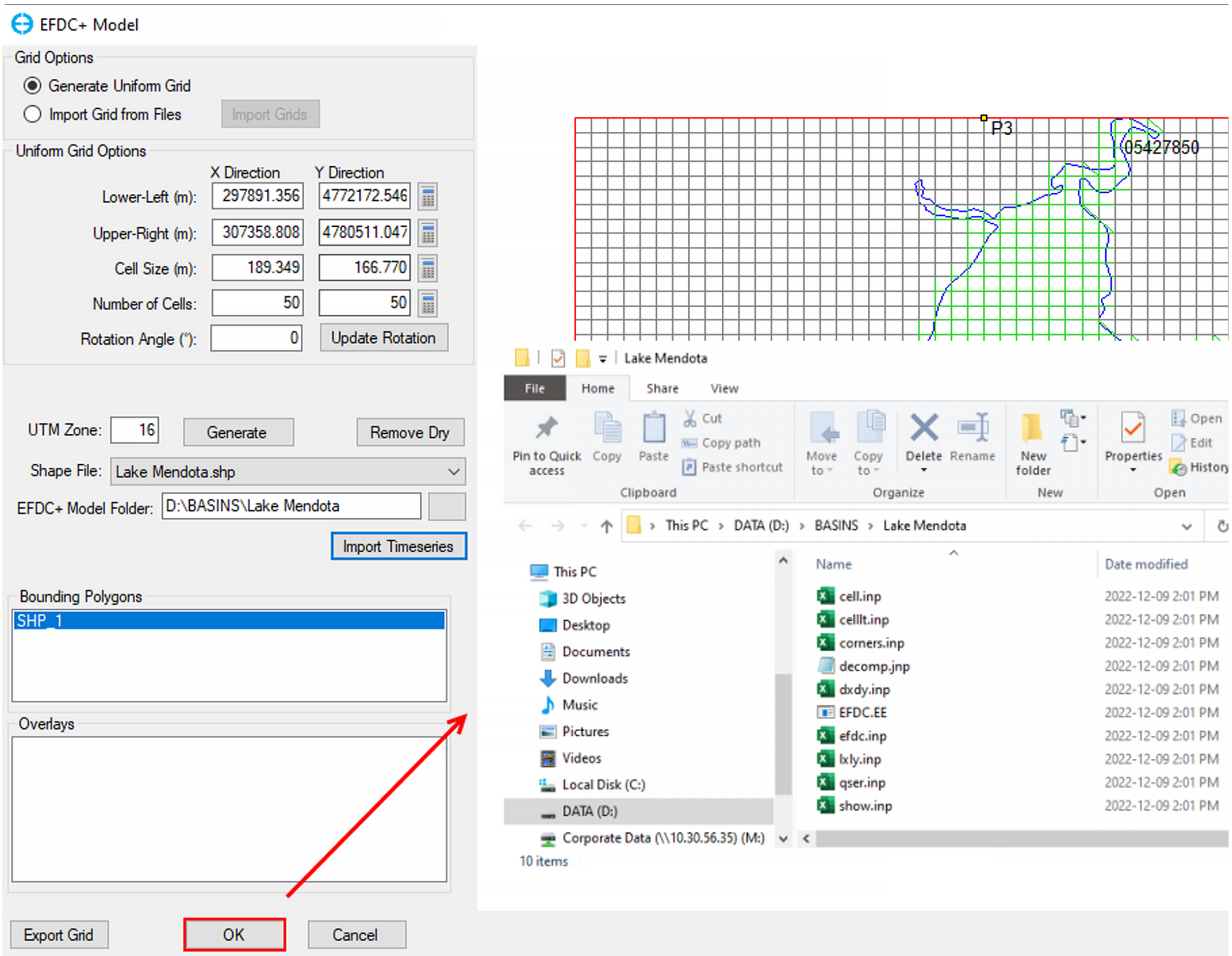

Figure 4.6: Folder with the EFDC+ Model of Lake Mendota.

It is also possible to export the grid to Grid+ format (*.gpc,*.cvl) and Delft3D RGF Grid (*.grd) (Figure 4.7).

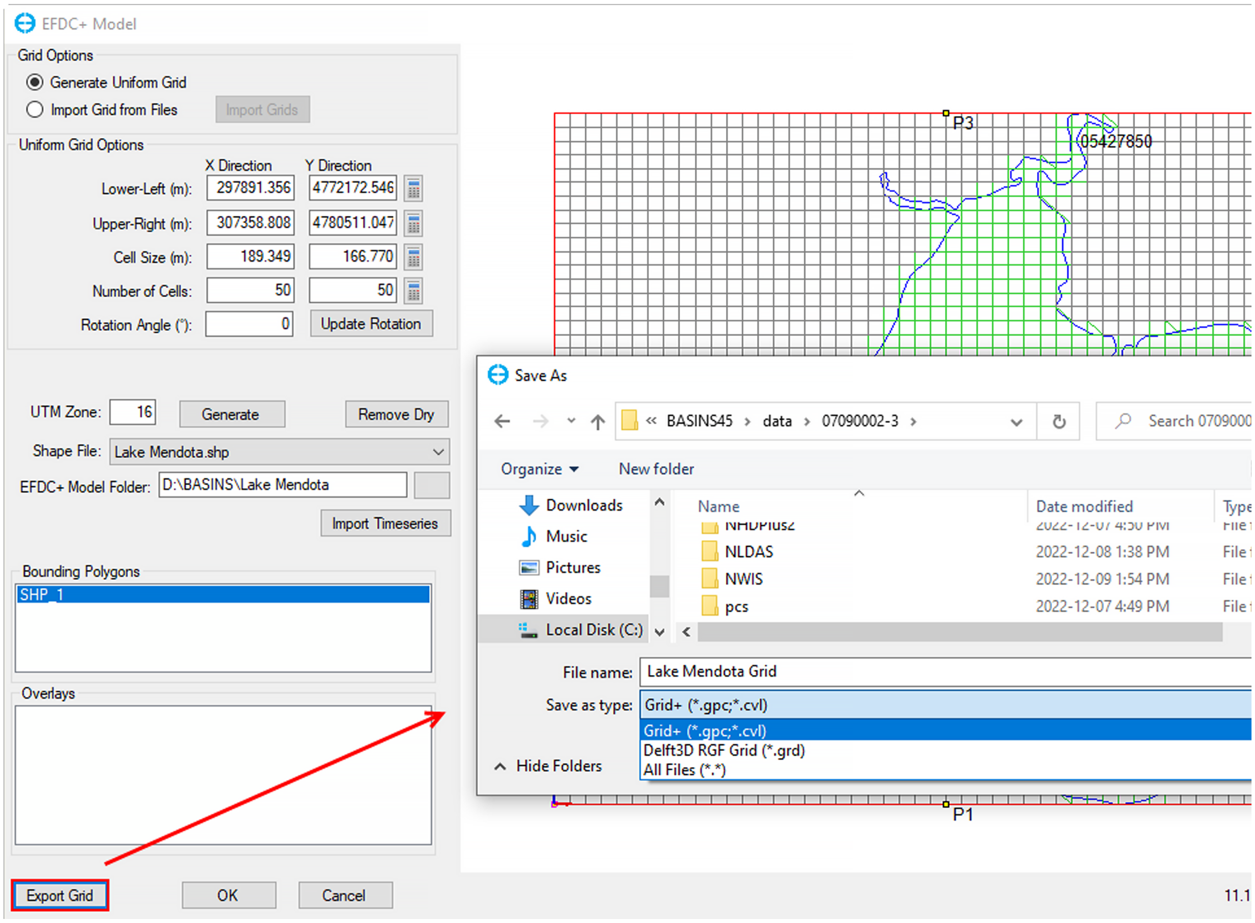

Figure 4.7: Export grid

Chapter 5

Build and Run Model with EE

Based on the EFDC+ model input files generated using the EEMS BASINS Plugin and BASINS programs from the previous sections, we will build and run a hydrodynamic model of Lake Mendota using EEMS in this section.

5.1. Open Model in EE

- Double-click the EE11.6 icon on your Desktop.
- Open the Lake Mendota model that was created in the previous section (Figure 5.1).

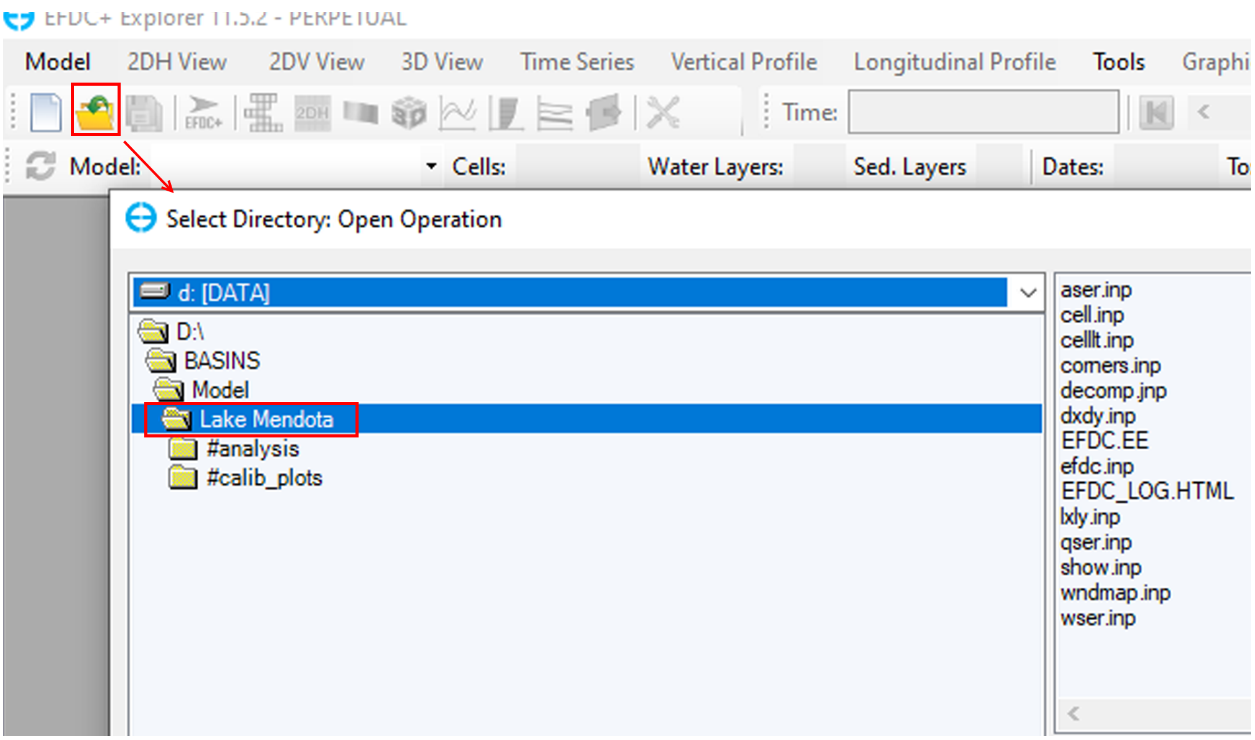

Figure 5.1: Open Lake Mendota model

5.2. Assign Model Bathymetry

This section will show how to assign bathymetry for the model domain. For this exercise, we have provided the bathymetry data for this location in a text file with X, Y, and Z values.

In the *Model Control* form (Figure 5.2):

1. Expand the *Initial Conditions* tab on the left-hand side.
2. RMC on *Bathymetry*.
3. The *Bathymetry* form will appear; click on the *Assign* button to assign model bathymetry.

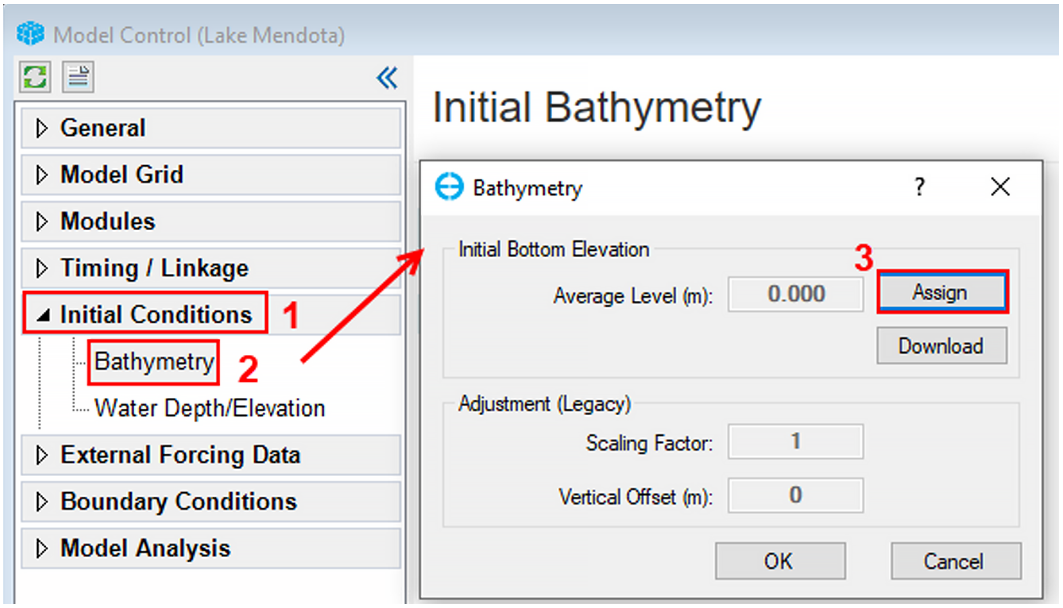

Figure 5.2: Assign bathymetry(1)

After clicking on *Assign*, the *Apply Cell Properties via Polygons: Bottom Elevation* form will appear. In this form (Figure 5.3):

1. In the *Values to Set* frame, select *Scatter (XYZ) data*.
2. In the *Data Files* frame, click on the *Add File* button to browse to the bathymetry data file named “Bottom Elevation.xyz” in the "Example Data - Lake Mendota" folder.
3. Click on *Apply Defined Conditions* to assign model bathymetry. This sets the bottom elevation data from the bathymetry file to the model.

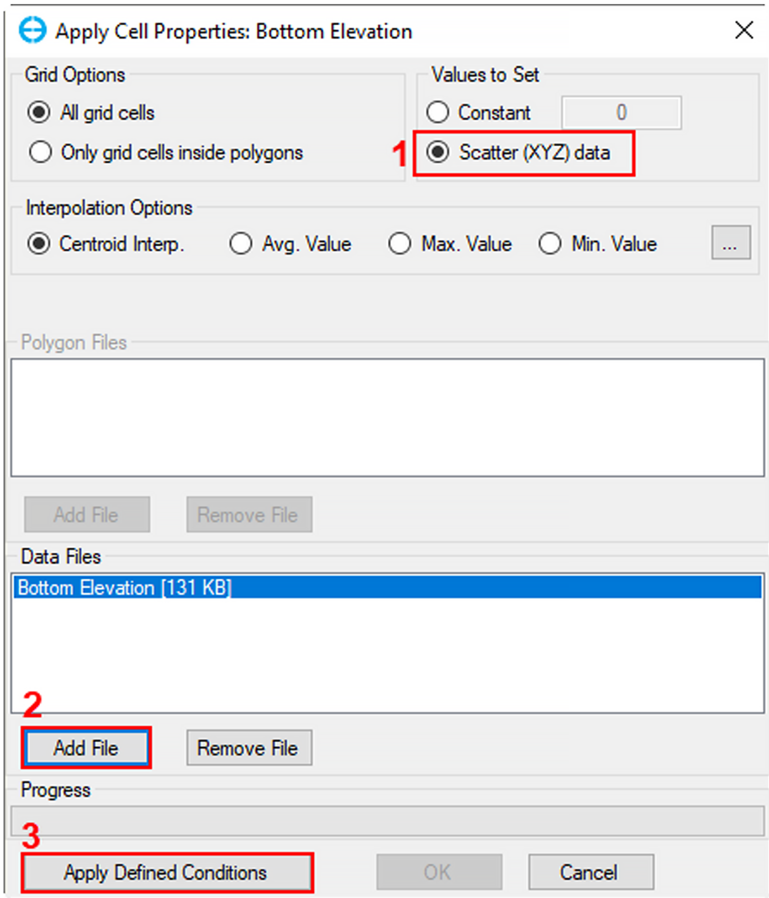

Figure 5.3: Assign bathymetry(2)

After assigning model bathymetry, a pop-up will appear to indicate the number of assigned cells and the assigned time (Figure 5.4). Click *OK* on this and all forms to close forms and go back to the *Model Control* form.

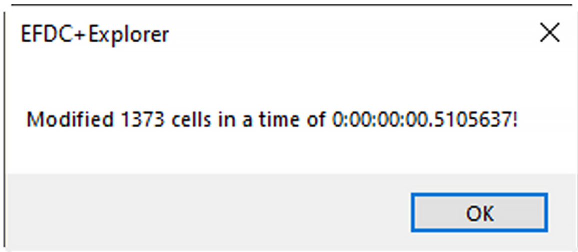

Figure 5.4: Assign bathymetry(3)

5.3. Assign Water Elevation

The initial model condition of water surface elevation can be assigned by following steps.

1. In the *Model Control*, under the *Initial Conditions*, RMC on the *Water Depth/Elevation*, then the *Water Depth/Surface Elevation* form will appear (Figure 5.5).
2. Click the *Assign Elevation* button to assign water surface elevation, then the *Apply Cell Properties: Water Surface Elevations* window will pop up.

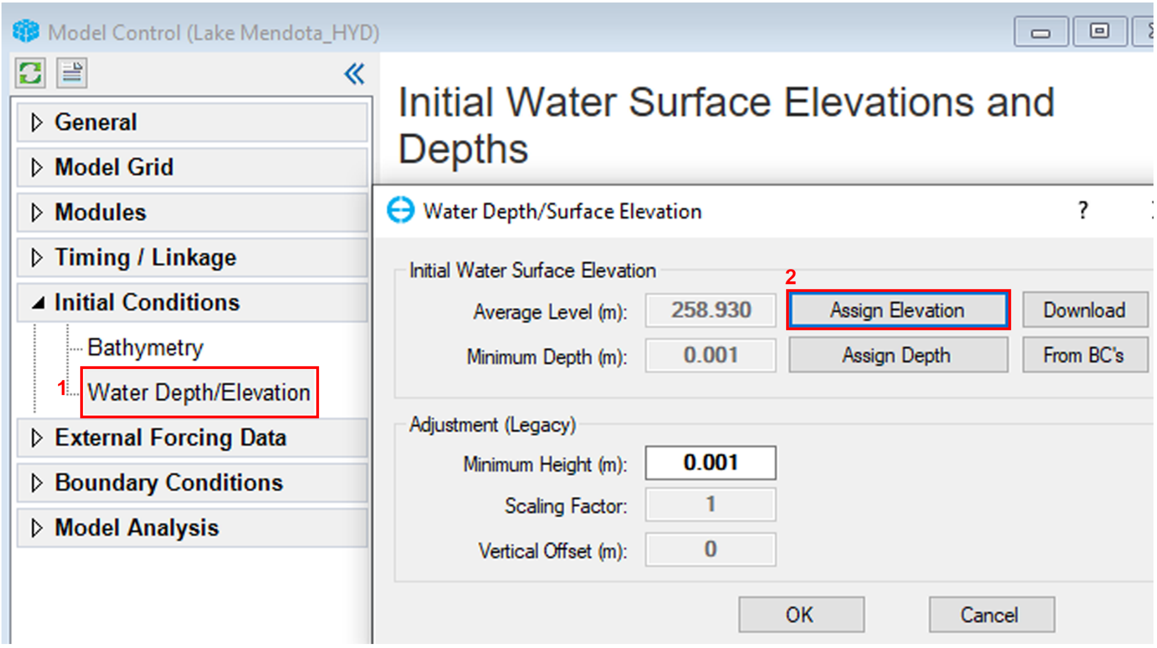

Figure 5.5: Assign water surface elevation

In the *Apply Cell Properties* window, follow the steps below (Figure 5.6).

1. In the *Grid Options*, select *All grid cells*.
2. In the *Values to Set* frame, select *Scatter (XYZ) data*.
3. In the *Data Files* frame, click on the *Add File* button to browse to the water elevation data file name “Water Elevation.xyz” in the "Example Data - Lake Mendota" folder.
4. Click *Apply Defined Conditions* to assign water surface elevations.

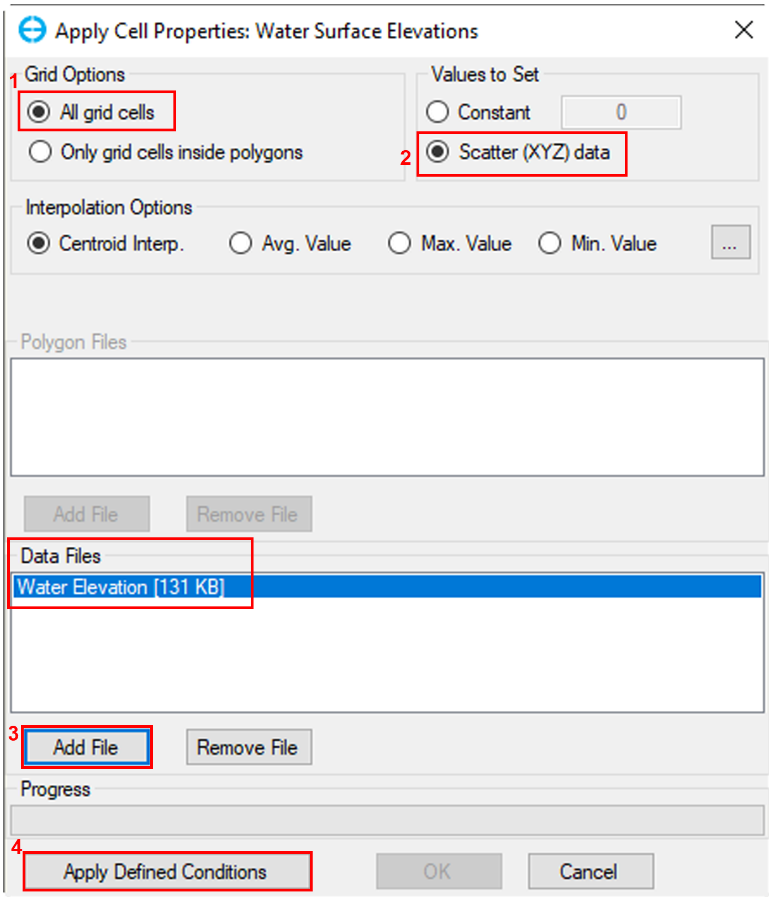

Figure 5.6: Apply Cell Properties: Water Surface Elevations

The pop-up message will appear to confirm the modification of the cells (Figure 5.7).

- Click *OK* to close the popup message and all windows and return to the *Model Control* form.

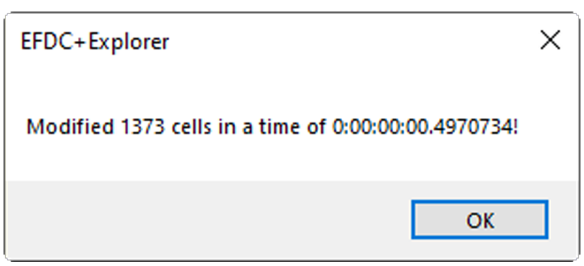

Figure 5.7: Report of number of assigned cells for initial water surface elevation

- After checking the initial water surface elevation, save the model.

5.4. External Forcing Data

5.4.1 Assign Outflow

In the following steps (Figure 5.8), we will assign one flow time series as outflow data.

1. Expand the *External Forcing Data* tab, *Flow* tab shows 5 flow data time series imported from BASINS.
2. RMC on the **OBSERVED 05428500 FLOW** data and select *Edit Data Series*, the *Boundary Data Series* will pop up.
3. In the *Boundary Data Series* form, the *Data Series* includes three columns *Time*, *Layer 1* and *Total*. LMC on the *Layer 1* column, the column will be highlighted.
4. We will multiply the *Layer 1* column values by -1 to define this flow time series as outflows leaving the model domain. In the toolbar above the time series table, enter "-1" into the box, and click the multiplier icon.
5. Click *OK* to close the *Boundary Data Series* window.
6. Save the model, again.

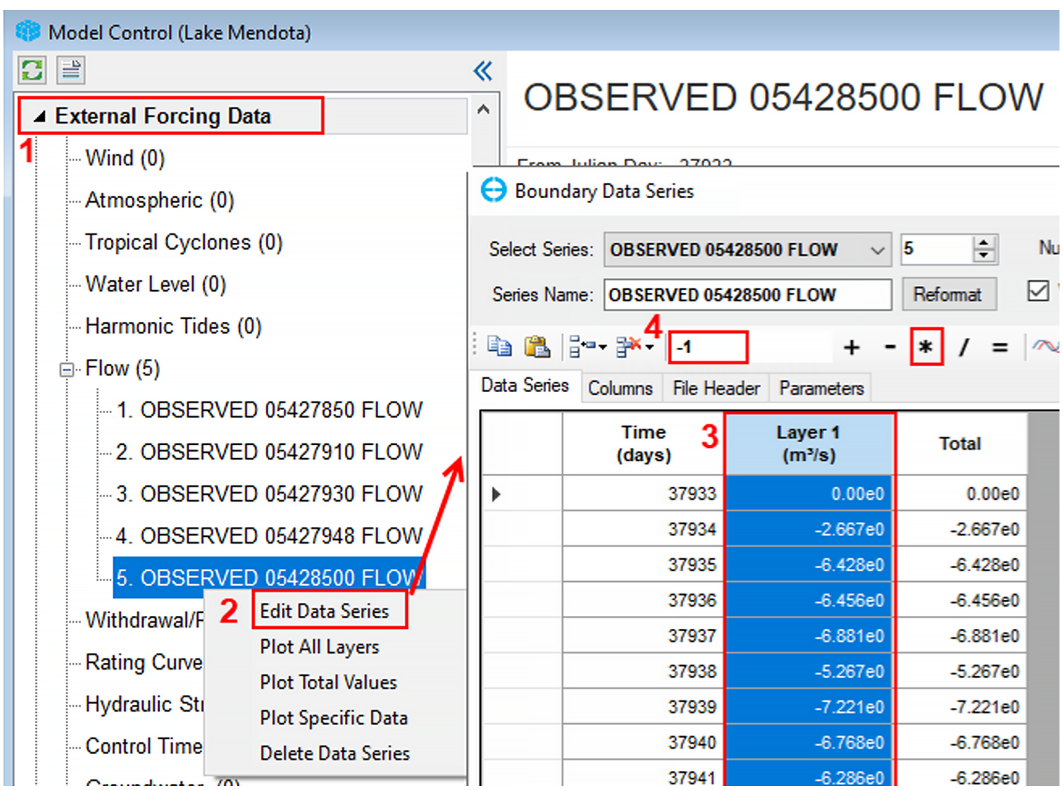

Figure 5.8: Assign Outflow

5.4.2 Add Ungaged Flow

Our initial testing suggests that Lake Mendota includes not only the flow from these four USGS gages but also the flow from some ungaged areas and local catchments. To account for those flows, we created an ungaged flow timeseries. The following steps describe the process to import the pre-defined ungaged flow data(Figure 5.9).

1. Expand *External Forcing Data* tab, RMC on *Flow* and select *Add New Data Series*.
2. The *Boundary Data Series* will pop up, click on *Add New* button.
3. Then click on *Import data from file* button, then the *ASCII Data Import* tab pops-up (Figure 5.10)

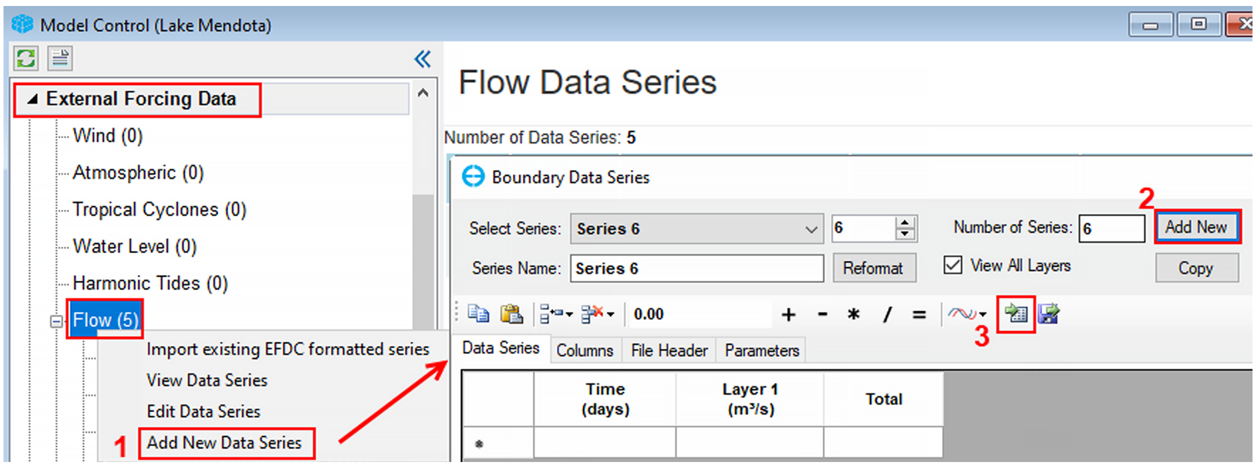

Figure 5.9: Add New Data Series

4. Click on *Browse* button to browse to the ungaged flow data *Ungaged flow.dat* in the *Example Data - Lake Mendota\Flow* folder.
5. Select the *Data Format* as *EE DAT or WQ Data*.
6. For *Import Settings*, select *Header Rows to Skip* is 1, *Data Base Date* is 1900-01-01 and *Import from Data Column* is 2.
7. Click *OK* to import data from the file.
8. For the *Import Data* window popped up, enter "-1" and click *OK*.

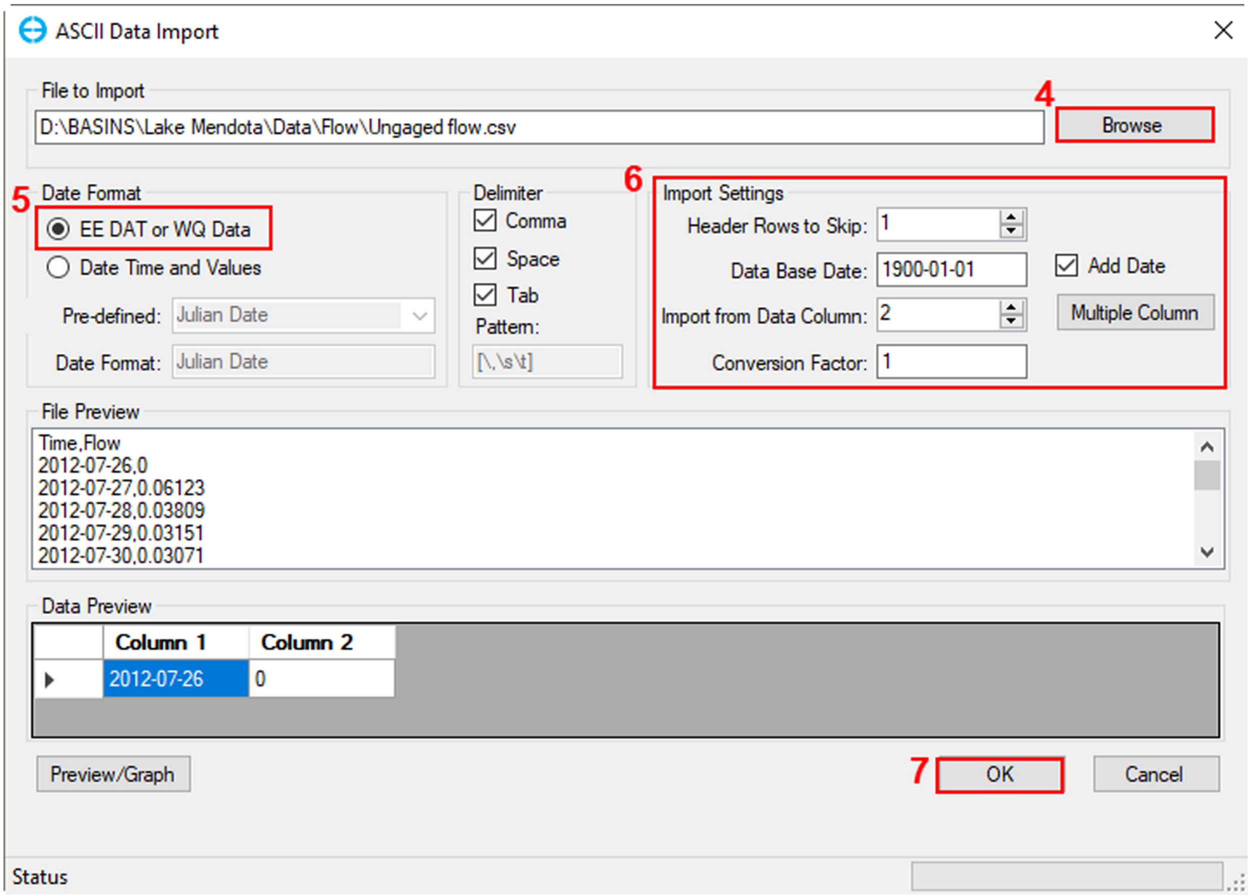

Figure 5.10: Import ungaged flow

After importing the ungaged flow, we will edit the location of the ungaged station (Figure 5.11).

1. In *Boundary Data Series* window, click *Parameters*
2. Enter the value of the parameter as shown in Table 5.1
3. After completing the parameter values, click *OK* to close the *Boundary Data Series* window.
4. Save the model, again.

Table 5.1: Parameters of ungaged flow station

Parameter	Value
Longitude (deg.)	-89.42597
Latitude (deg.)	43.1141
X (m)	0
Y (m)	0
Time Scale (sec.)	86400
Time Offset (sec.)	0
Invalid Flag	-999

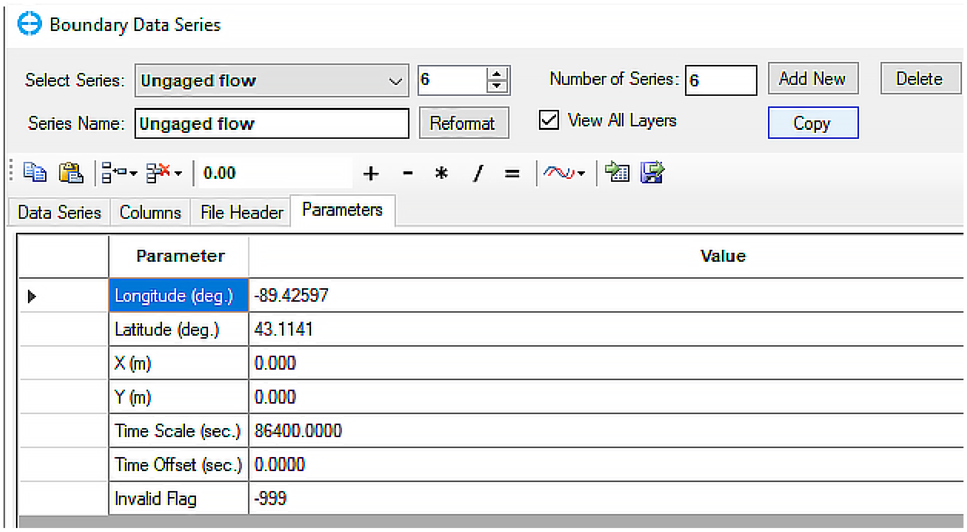

Figure 5.11: Parameters of ungaged flow

5.4.3 Add Wind Time Series

The steps in Figure 5.12, Figure 5.13, and Figure 5.14 show how to import wind time series for the model.

1. Expanding the *External Forcing Data* tab, and RMC on *Wind*, select *Add New Data Series*.
2. In the *Boundary Data Series* window, click on *Add New* button to add a new data series.
3. Click on *Import data from file* button.

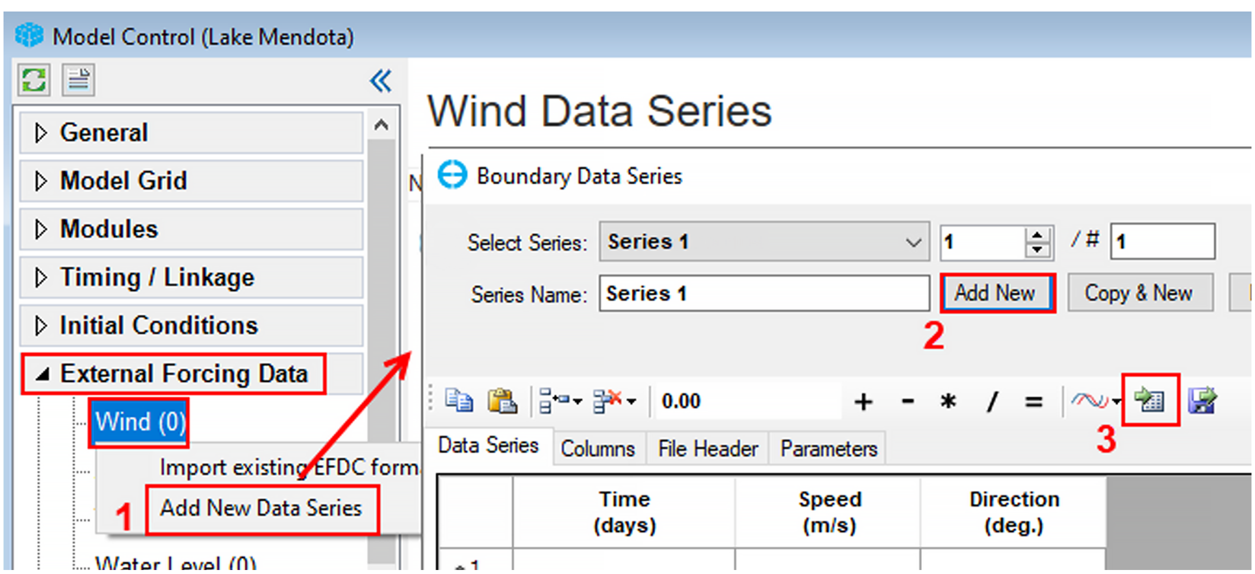

Figure 5.12: Import wind boundary time series (1)

4. In the *ASCII Data Import* window that pops up, browse to the *"Dane Country Airport.csv"*

file in the *Wind* data folder of *Example Data - Lake Mendota* folder.

5. Select the data format as *EE DAT or WQ Data*.
6. Check the *Comma* and *Tab* box in *Delimiter* frame (Uncheck the *Space* box).
7. In the *Import Settings*, click the *Multiple Column* button.
8. In the *Import Multiple Columns from File* window that pops up, select *Speed* as Data Column 2 and *Direction* as Data Column 3. The *Data Preview* shows the preview of the imported data.
9. Click *OK* to close the *Import Multiple Columns from File* window.
10. Click *OK* in the *ASCII Data Import* window, and the wind data import process will complete.

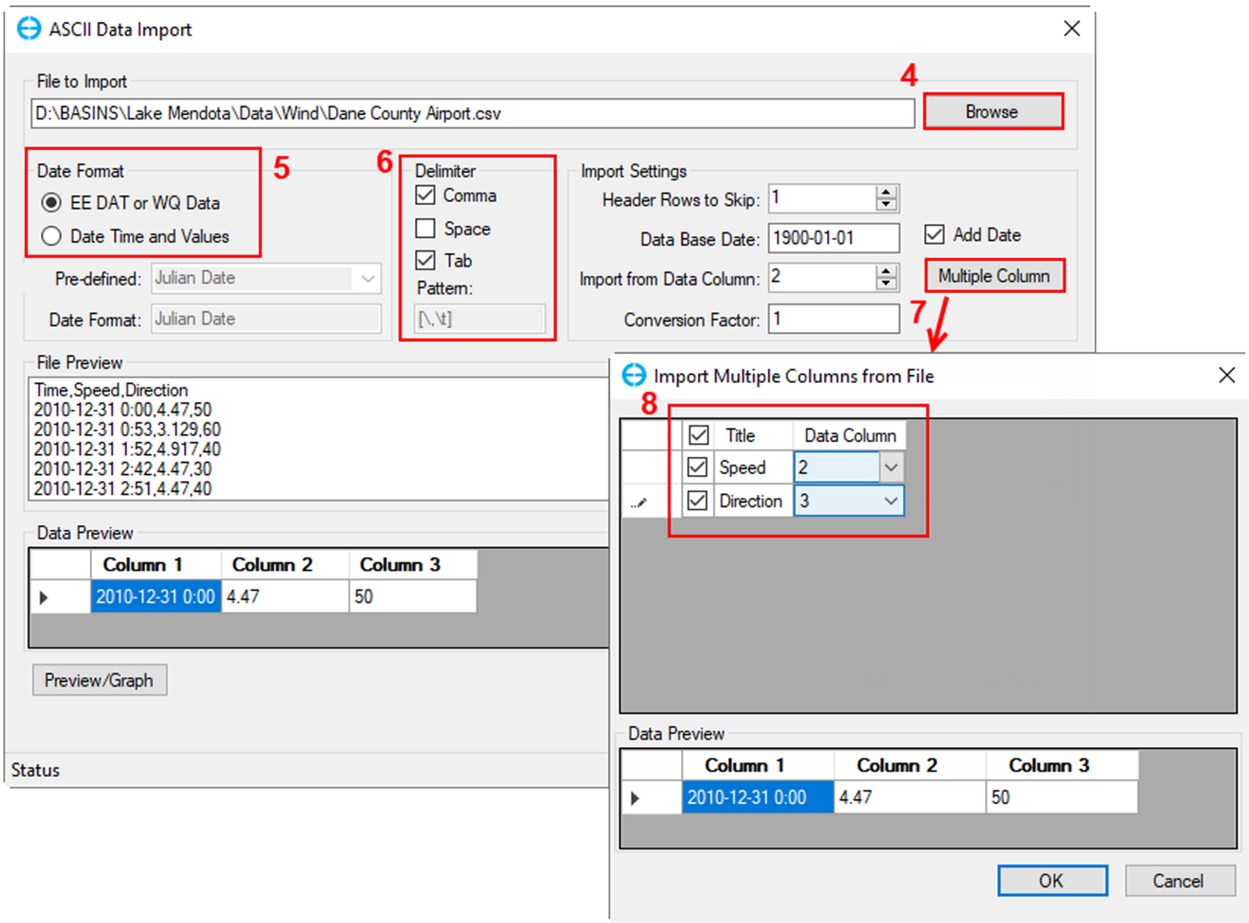

Figure 5.13: Import wind boundary time series (2)

11. Once the data import is done, back to *Boundary Data Series* window, enter the *Series Name* as *Dane County Airport*.
12. In *Parameters* sub-tab, set up the parameters of the boundary data as Figure 5.14.
13. After entering the parameter values, click *OK* to close the *Boundary Data Series* window.
14. Save the model.

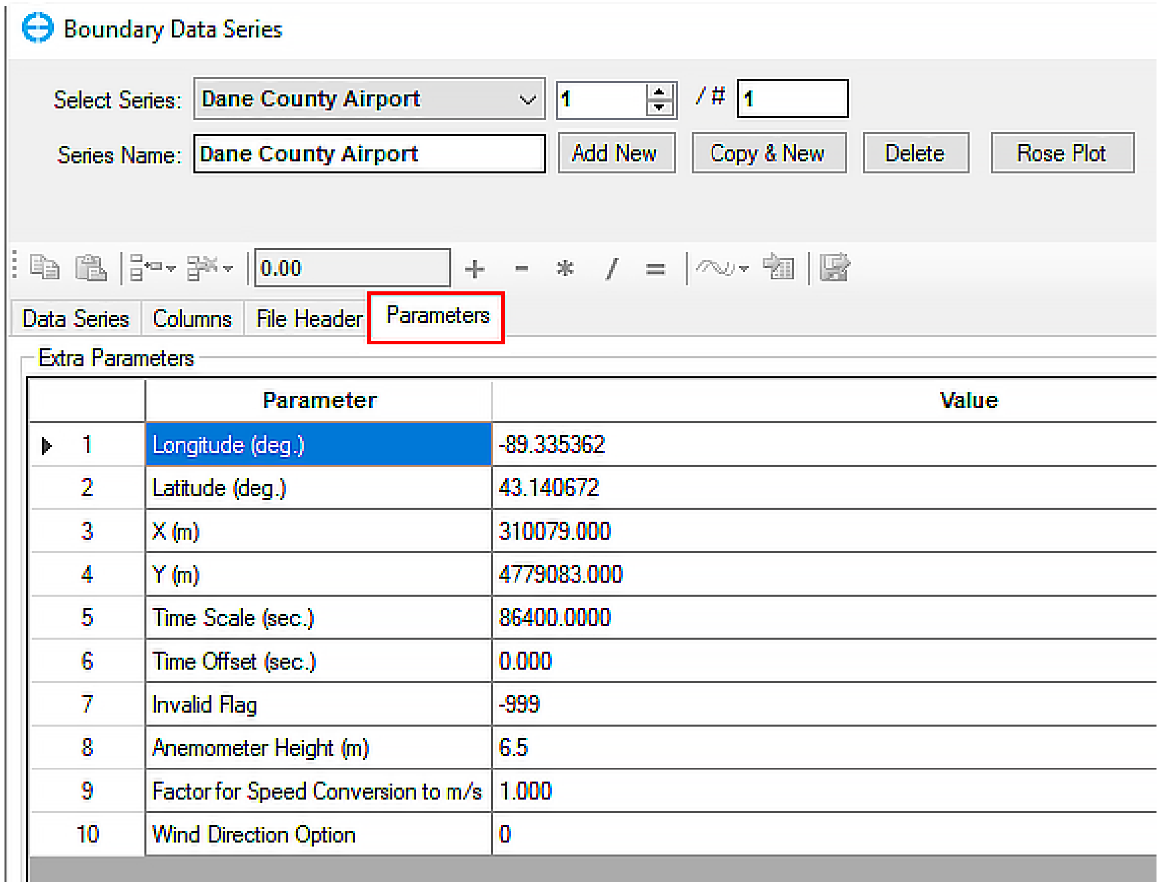

Figure 5.14: Parameters of Dane County Airport

5.4.4 Add Atmospheric Time Series

We will now import atmospheric data into the model.

1. In the *Model Control form*, expand the *External Forcing Data* tab. RMC on the *Atmospheric*, and click *Add New Data Series* (Figure 5.15).
2. The *Boundary Data Series* window will pop up, click *Add New button*.
3. Click the *Import Data from File* icon to open the ASCII Data Import form.

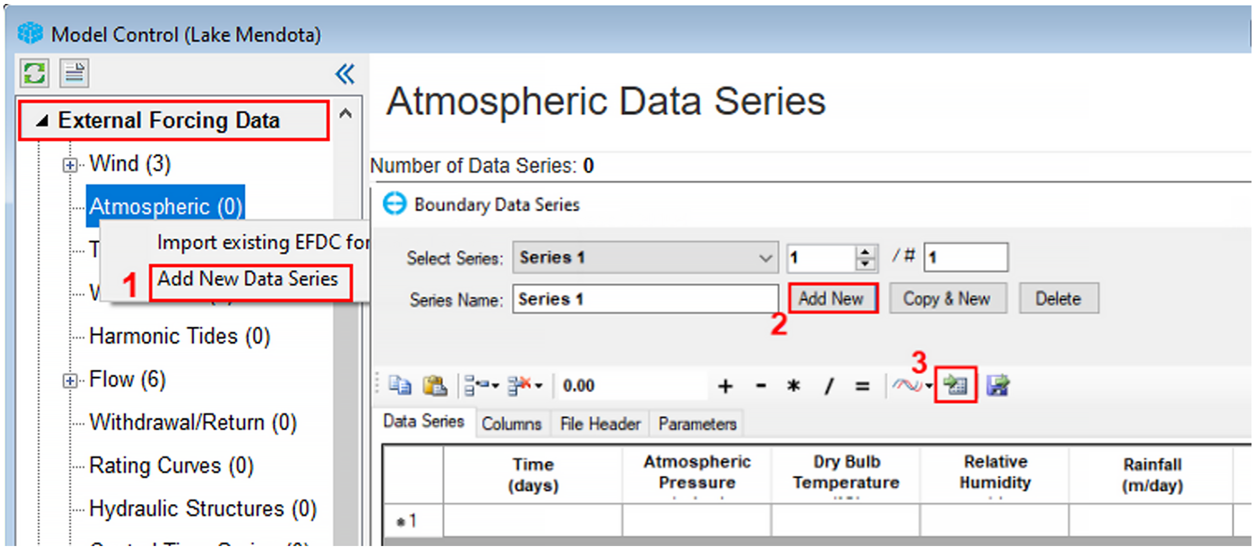

Figure 5.15: Add Atmospheric Data Series (1)

In the *ASCII Data Import* window (Figure 5.16),

4. Click the *Browse* button and browse to the file *Dane County Airport.csv* in the *Atmospheric* data folder of *Example Data - Lake Mendota* folder.
5. In the *Date Format*, select *EE DAT or WQ* option.
6. Select *Delimiter* as *Comma* and *Tab* (Uncheck the *Space* box).
7. In *Import Settings*, set *Header Rows to Skip* as 1 and select *Data Base Date* as 1900-01-01 and click the *Multiple Column* button.
8. Select data fields to corresponding columns shown in Figure 5.16, and click *OK* to close the *Import Multiple Columns from File* window
9. Click *OK* in the *ASCII Data Import* window to start importing data to the model.

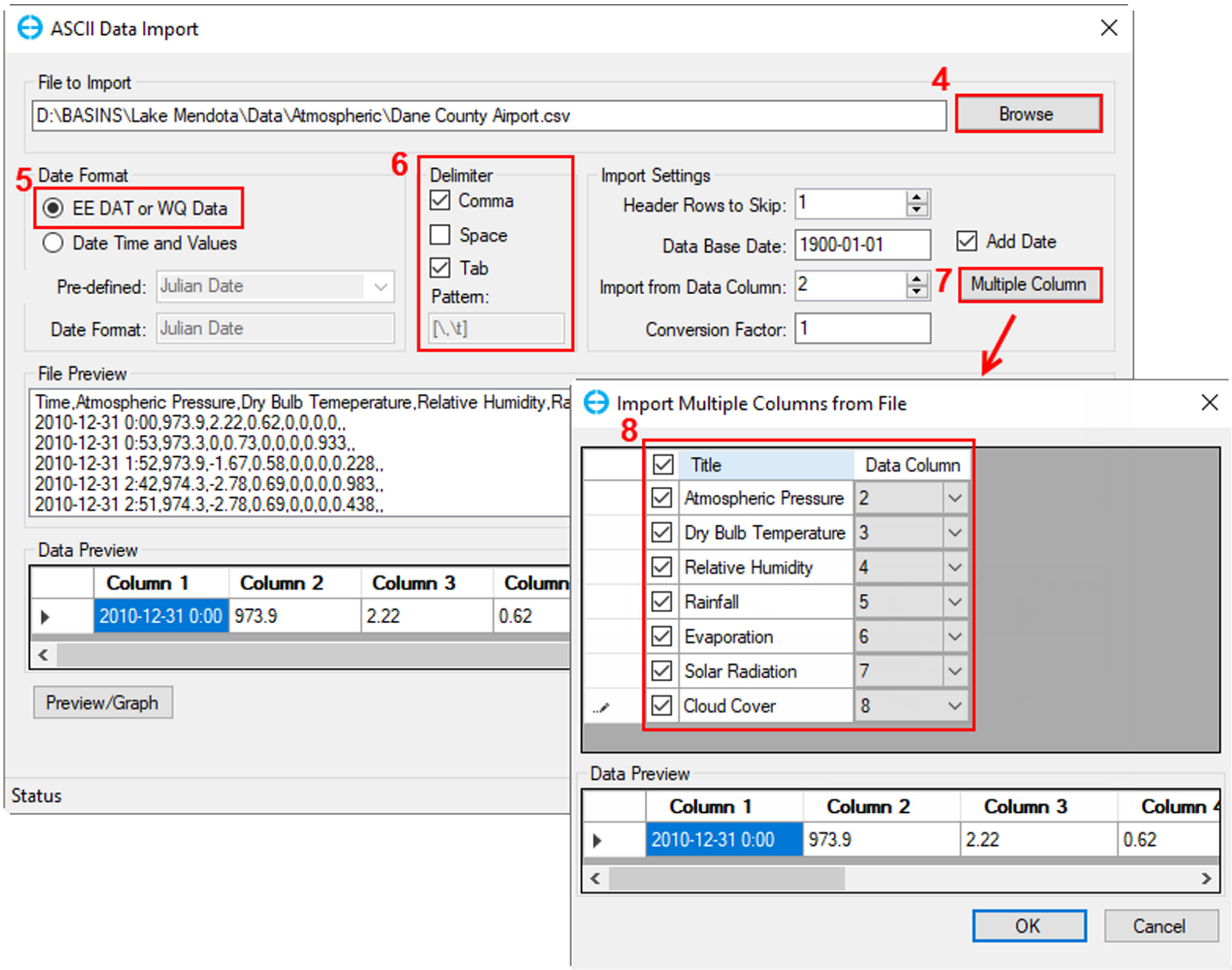

Figure 5.16: Add Atmospheric Data Series (2)

Going back to the Boundary Data Series window,

1. Rename *Series Name* as Dane County Airport.
2. Click the *Parameter* tab (Figure 5.17).
3. Update the parameters of the atmospheric station.
4. Click *OK* to close the *Boundary Data Series* window.
5. Save the model.

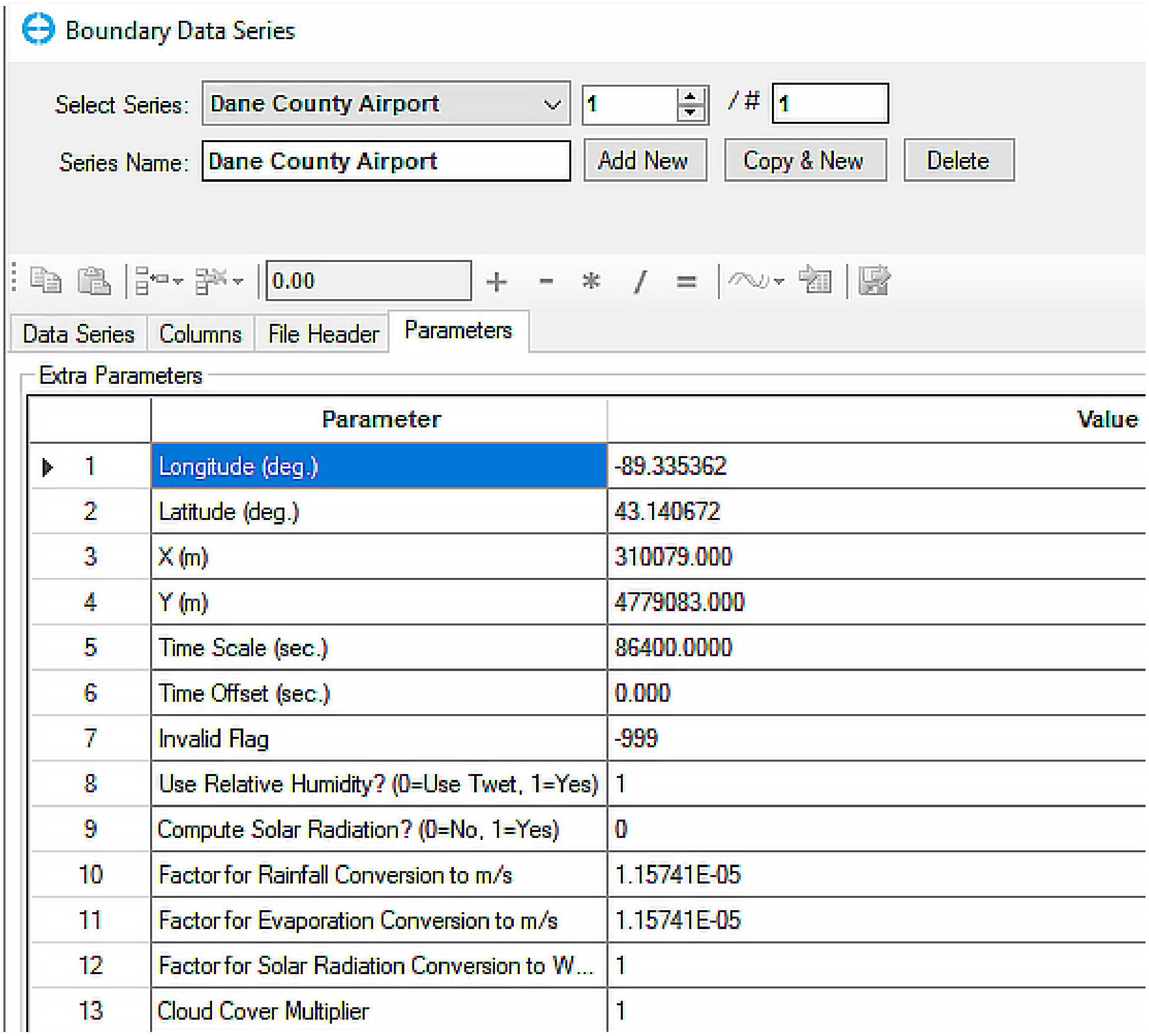

Figure 5.17: Parameters of Dane County Airport meteorological station

5.4.5 Add Groundwater Time Series

We will now add groundwater time series data to the model.

1. In the *Model Control form*, expand the *External Forcing Data* tab. RMC on the *Groundwater*, and click *Add New Data Series*, as shown in Figure 5.18.
2. The *Boundary Data Series* window will pop up, click *Add New button*
3. Click the *Import Data from File* icon to open the ASCII Data Import form

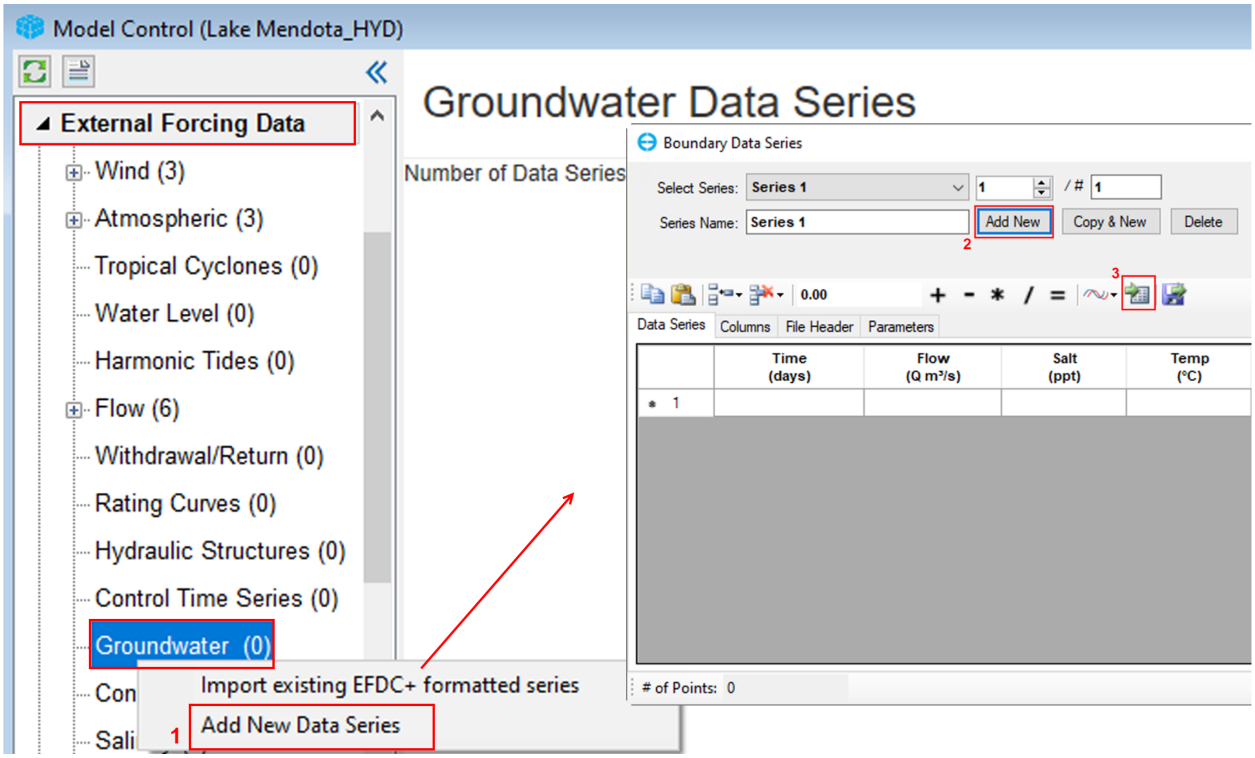

Figure 5.18: Add Groundwater Data Series(1)

4. In the *ASCII Data Import* window that pops up, browse to the *"GroundWater.dat"* file in the *Groundwater* data folder of *Example Data - Lake Mendota* folder.
5. Select the data format as *EE Dat or WQ Data*.
6. Check the *Delimiter* options for *Comma* and *Tab* box (Uncheck the *Space* box).
7. In the *Import Settings*
    - *Header Rows to Skip*: 7
    - *Data Base Date*: 1900-01-01
    - *Import from Data Column*: 2
    - *Conversion Factor*: 1
8. Click *OK* to import the data and close the *ASCII Data Import* window.
9. Click *OK* in *Boundary Data Series* window.
10. Save the model.

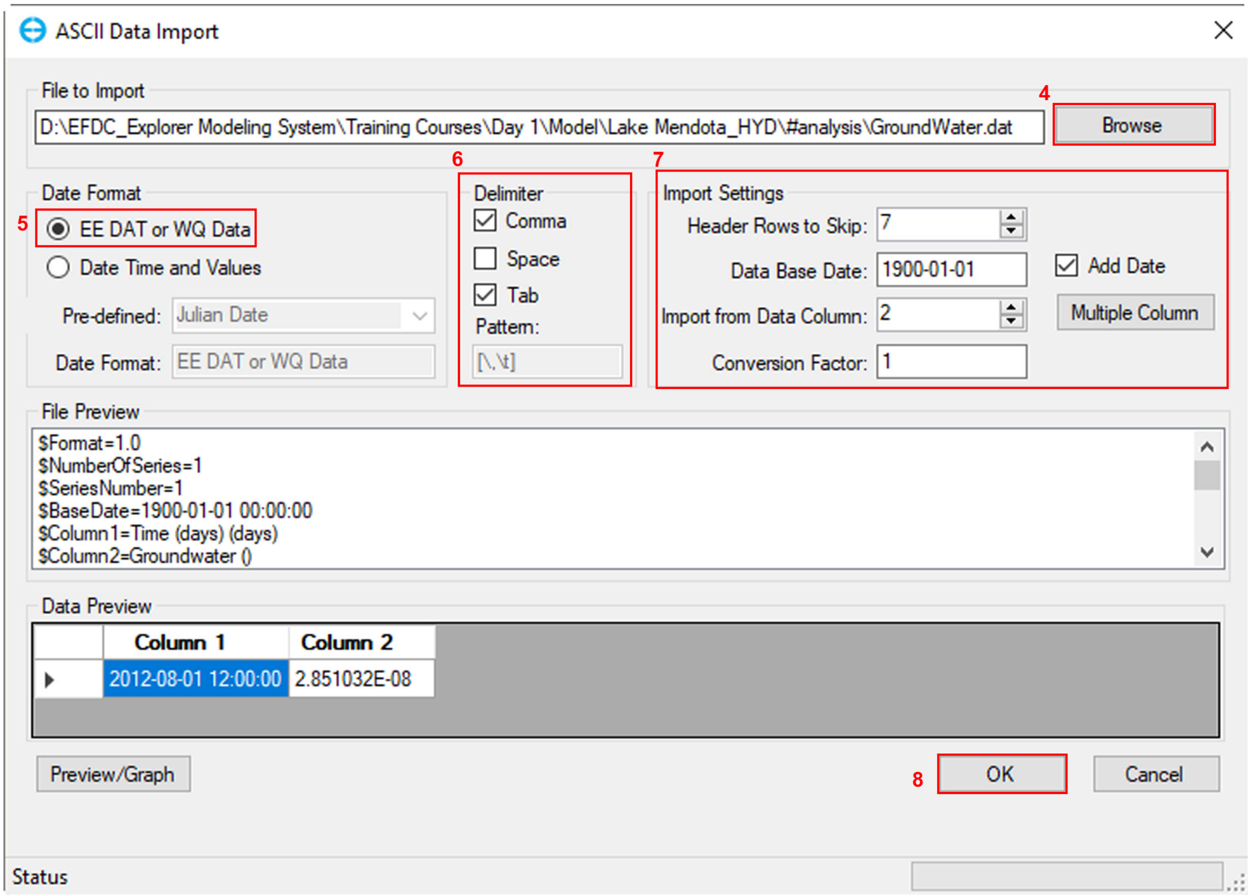

Figure 5.19: Add Ground Water Data Series(2)

In the *Model Control* form, you can view the summary of groundwater time series imported (Figure 5.20).

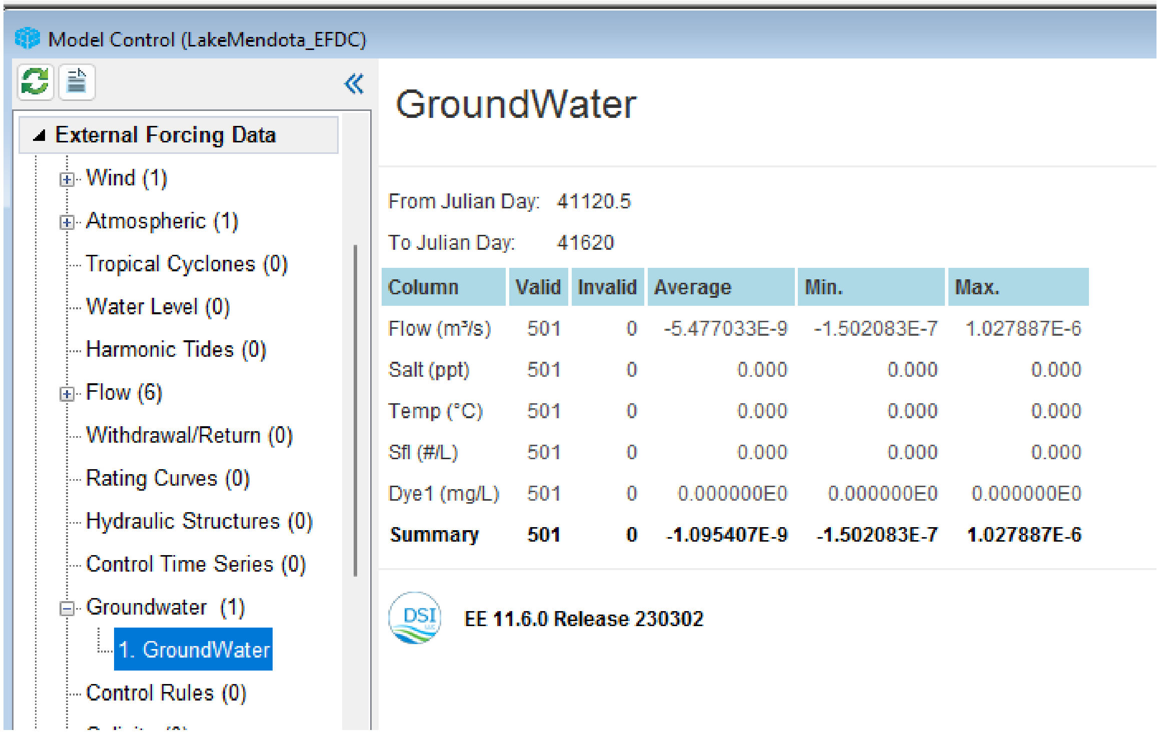

Figure 5.20: Groundwater summary

In the next steps, we will assign groundwater boundary conditions (Figure 5.21) to model grid cells.

1. In the *Model Control form*, expand the *Boundary Conditions* tab and RMC on the *Groundwater*, the *Groundwater* form will appear.
2. In the *Groundwater* window, select *Groundwater Interaction* option in the drop-down list.
3. Click *Apply Overlay* button.

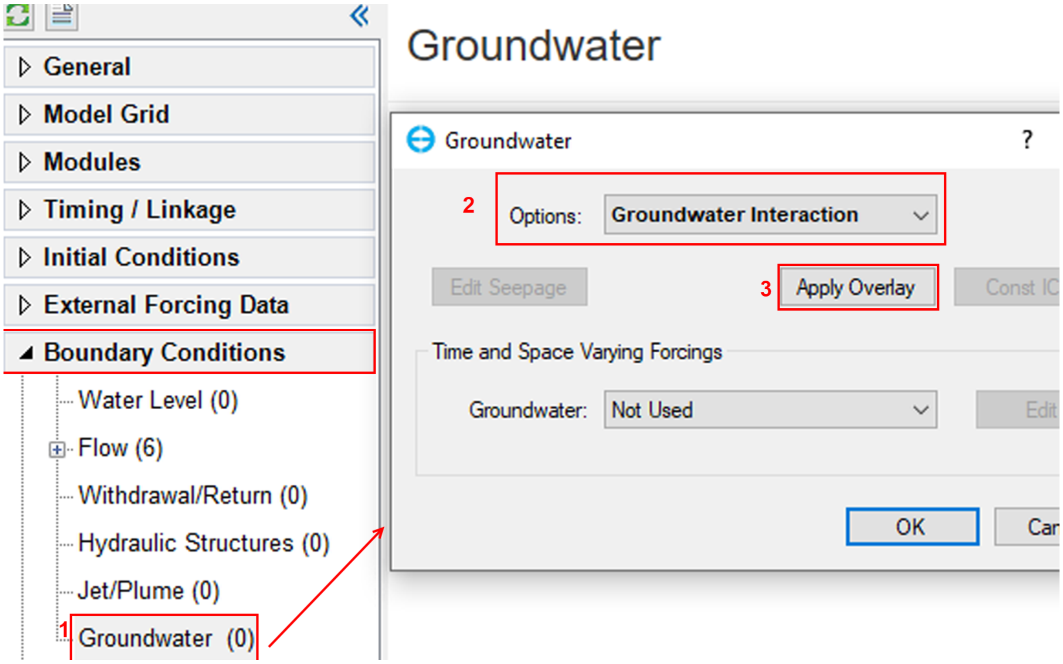

Figure 5.21: Groundwater interaction

In the *Apply Cell Properties: Groundwater* window popped up as shown in Figure 5.22,

4. Select the *Grid Options* as *All grid cells*.
5. Select the *Values to Set* as *GroundWater* class.
6. Click *Apply Defined Conditions* button.
7. Click *OK* to close the windows after the import process is done (the cell modification message will appear).
8. Save the model.

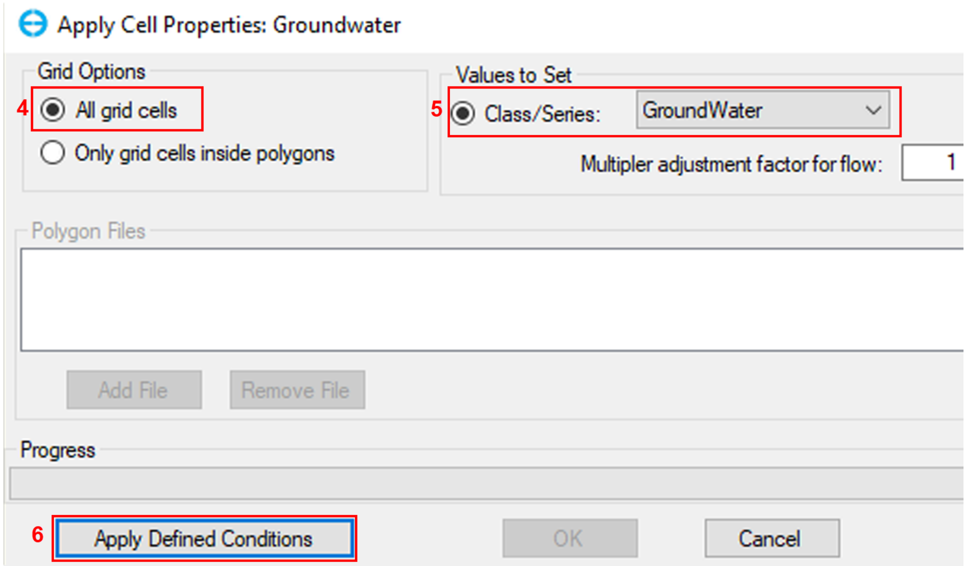

Figure 5.22: Apply Cell Properties Groundwater

5.4.6 Add Ungaged Flow Boundary Conditions

In the following steps, we will add a new boundary group (Figure 5.23) to the model cell to assign the ungaged flow data imported above.

1. Click the *2DH View* button from the main toolbar.
2. In *Layer Control*, click the light bulb, pointer, and pencil in the *Boundaries* layer to activate the editing mode of the layer.
3. Click *Select Object* icon under *Select Tool* button in the *2DH View* tool bar.
4. Click (LMC) a cell in the model domain to select (you may select any cell).
5. RMC on the selected model cell.
6. In the *Add BC Group* pop-up window, select *Boudary Group Type* as *Flow Boundary*, and name it as *Surface Balance Flow*.
7. Click *OK*, then the *Flow Boundary Conditions* window will pop up.

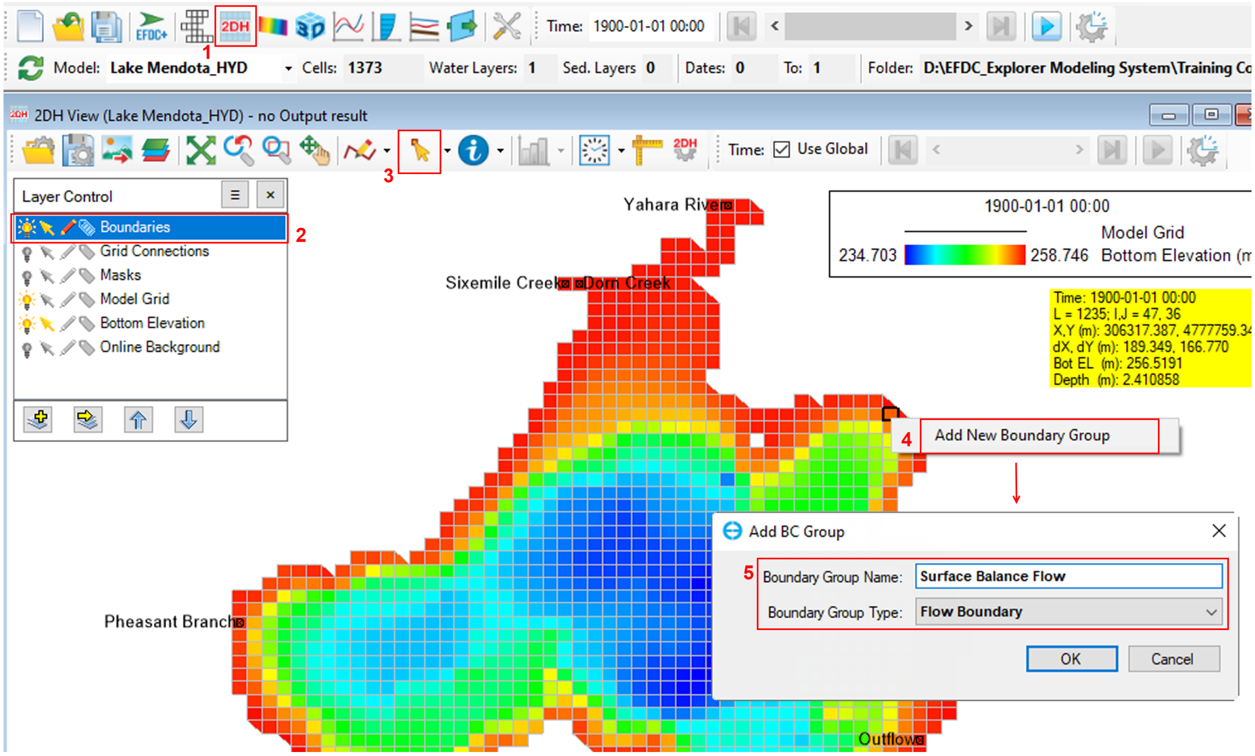

Figure 5.23: Add new boundary group

In the *Flow Boundary Conditions* window (Figure 5.24),

1. Select *Unaged flow* from the *Flow Time Series* drop-down list.
2. Click *OK* button to save the option and close the form.
3. In the *2DH View* window, see the *Surface Balance Flow* boundary is assigned to the selected cell.
4. Close the *2DH View* window.
5. Save the model.

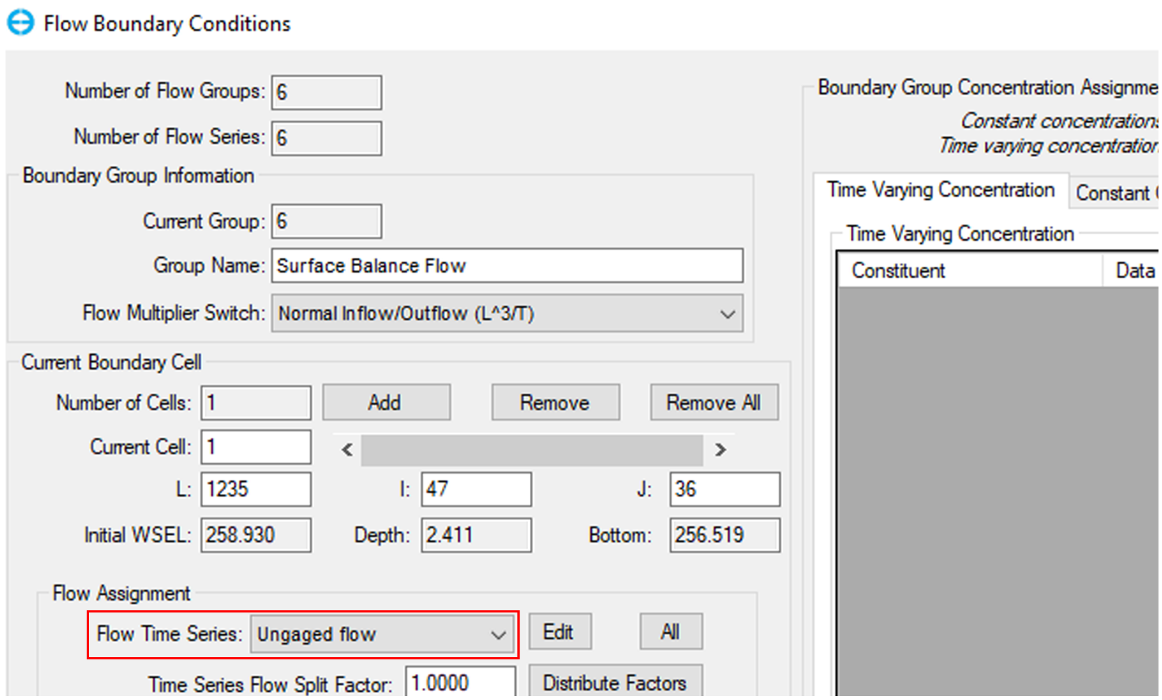

Figure 5.24: Assign surface balance boundary group

5.5. Hydrodynamic Settings

In the following steps, we will specify the hydrodynamic model parameters and setups.

5.5.1 Turbulence Options

From the *Model Control* form:

1. Go to the *Modules* tab on the left.
2. Expand the *Hydrodynamics* sub-tab.
3. RMC on *Turbulence* to open the *Turbulence Options* form.
4. In the *Turbulence Options* form, set the *Turbulence Diffusion* parameters as shown in Figure 5.25 below.
5. In the *Turbulence Options* form, go to the *Turbulence Intensity* tab. Then, set the parameters as shown in Figure 5.26.
6. After completing the turbulence parameter settings, click *OK* to close the *Turbulence Options* form.
7. Save the model.

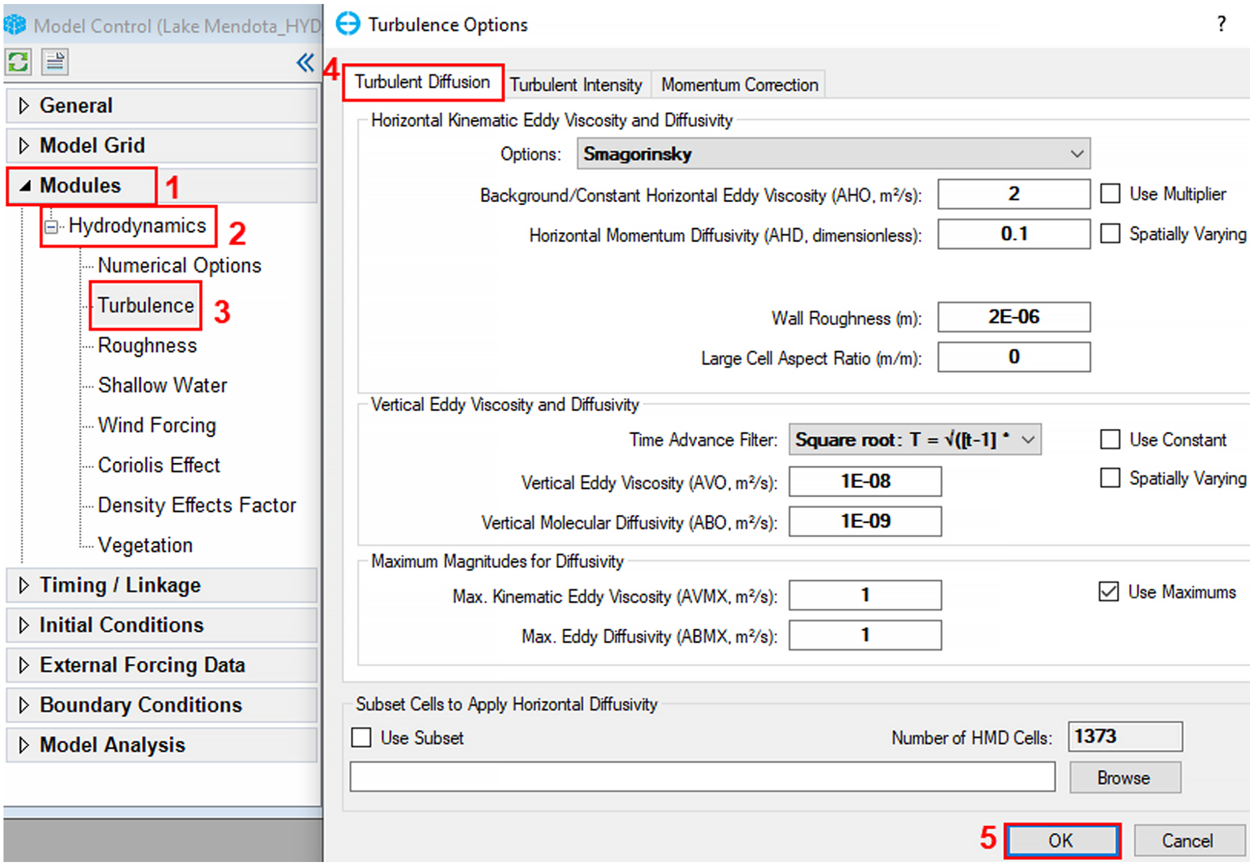

Figure 5.25: Turbulent Diffusion

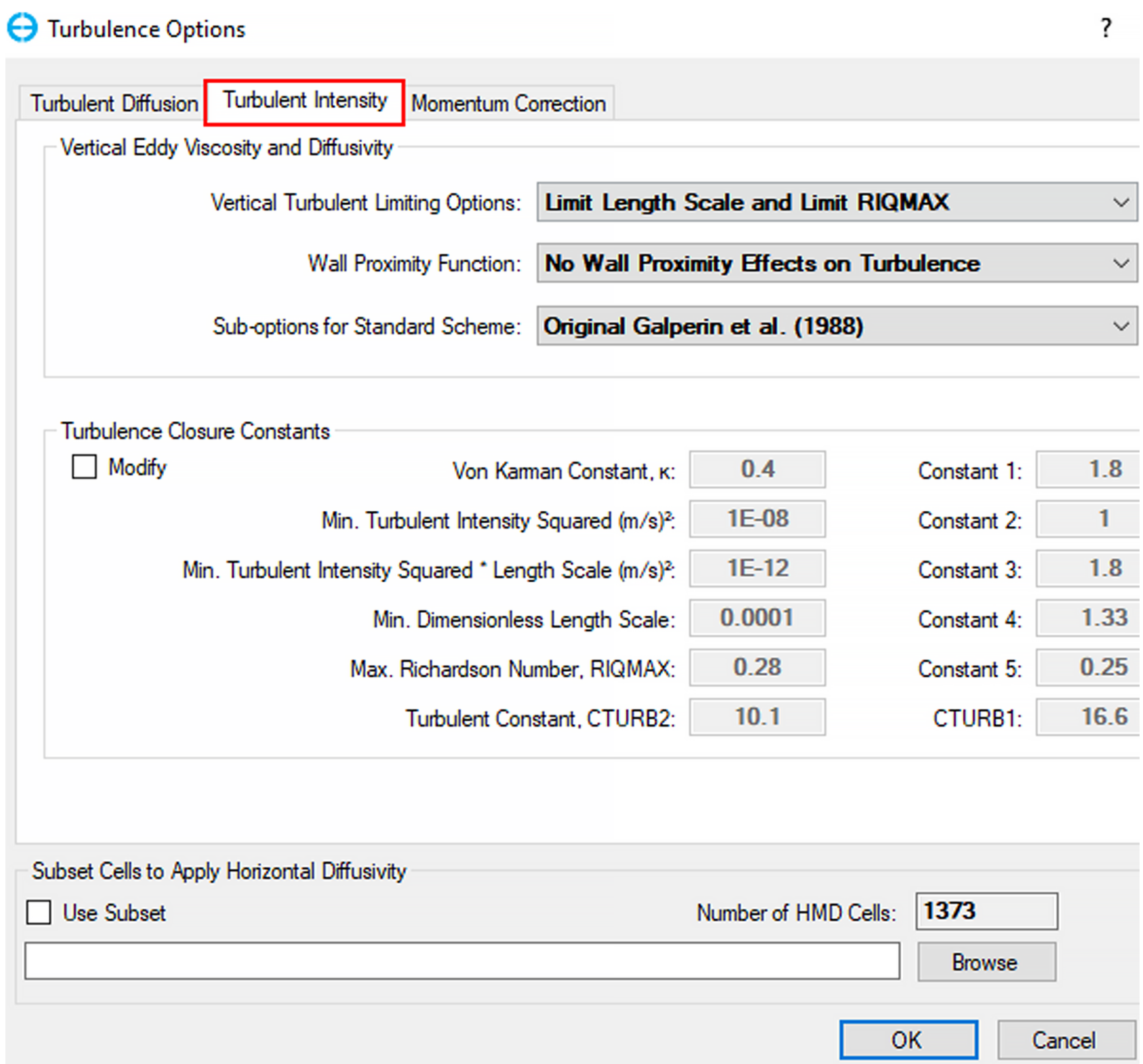

Figure 5.26: Turbulent Intensity

5.5.2 Roughness Settings

From the *Model Control* form (Figure 5.27):

1. Go to the *Modules* tab on the left of the *Model Control* form, expand the *Hydrodynamics* sub-tab, and RMC on *Roughness*.
2. In the *Bottom Roughness* window popped up, set *Drag Coefficient Approach* as *EFDC+ 10 and Greater* from the drop-down list.
3. Click the *Assign* button.
4. In the *Apply Cell Properties: Bottom Roughness* window popped up, for the *Values to Set* frame, select *Constant* and set the value to 0.005.
5. Click the *Apply Defined Conditions* button to assign the bottom roughness to all model cells.

6. Click *OK* to close the windows *Apply Cell Properties: Bottom Roughness* and *Bottom Roughness*.
7. Save the model.

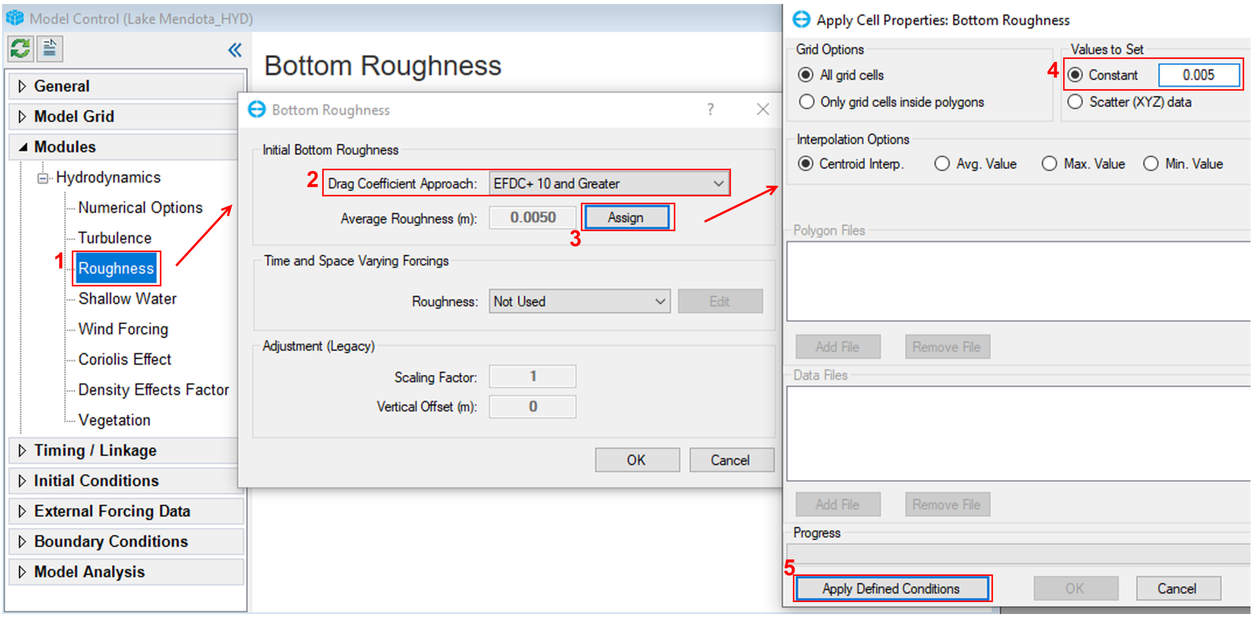

Figure 5.27: Applying Bottom Roughness

5.5.3 Shallow Water settings

From the *Model Control* form (Figure 5.28):

1. Go to the *Modules* tab, expand the *Hydrodynamics* sub-tab, and RMC on *Shallow Water*, then the *Shallow Water Options* window will appear.
2. Set the *Option* as *Use wetting/drying with non-linear iterations and face masking*.
3. Set the *Dry Depth* as 0.1 m.
4. Set the *Wet Depth* as 0.15 m.
5. Click *OK* to close the *Shallow Water Options* window.
6. Save the model.

This feature is required with fluctuating water levels that will leave some of the domain dry. Enable the wet and dry option is recommended for an area with tidal flats. Although it incurs a certain computational overhead, it is necessary to obtain accurate results since it sets the minimum water depth before a cell is taken out of calculation (*Dry Depth*) and also the water depth at which the cell hydrodynamics will be calculated (*Wet Depth*). The *Wet Depth* should always be greater than the *Dry Depth*.

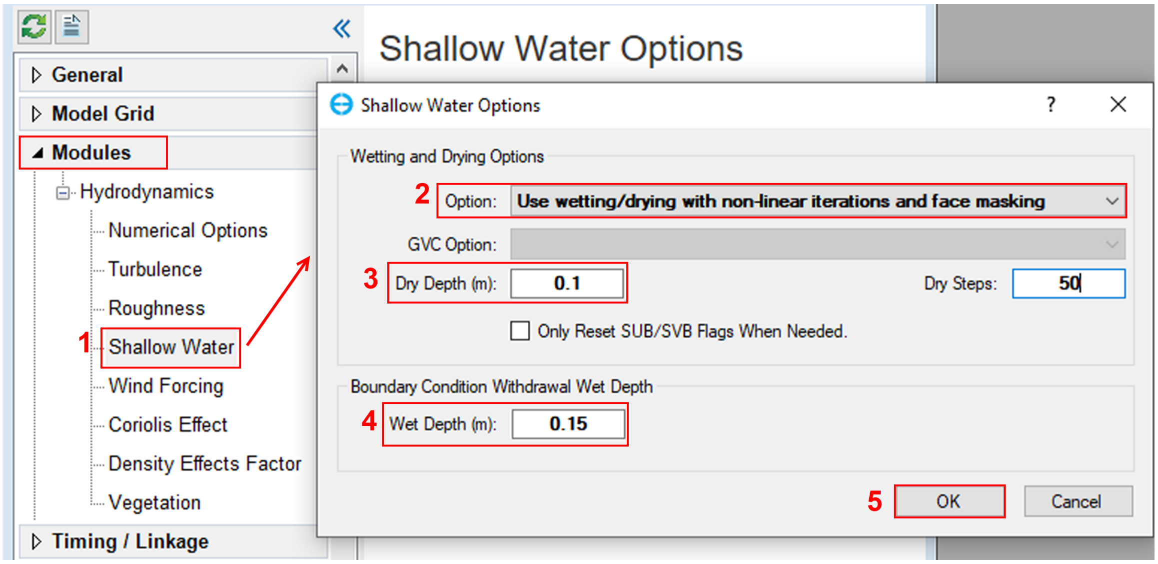

Figure 5.28: Hydrodynamic - Shallow water setting

5.6. Model Timing

Now, we are going to set the time period for the model simulation. Refer to the EE Knowledge Base on Set Timing frame for more details.

In EFDC+ Explorer, from the Model Control form, go to the Timing/Linkage tab and RMC on the Timing sub-tab, which will cause the Simulation Timing Options form to be displayed (refer to Figure 5.29). On this form, the Reference Date/Time is set as default. Time of Start is the number of Julian days after the base date of the simulation. The Number of Reference Periods is the total simulation time as a multiple of the reference periods. Time Step is the simulation time step. In the Mendota Lake example, the simulation time is set as shown Figure 5.29. For different models, the Dynamic Time Step Options the values settings may vary from those shown in Figure 5.29.

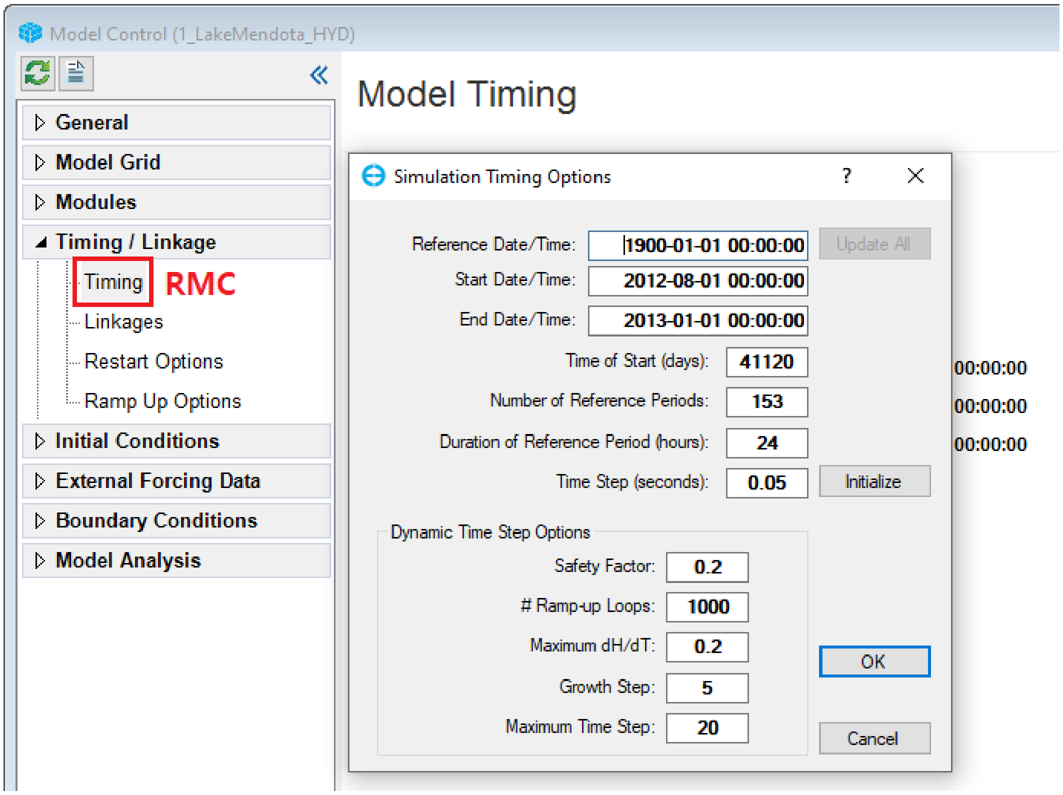

Figure 5.29: Simulation timing options

Now, we will specify the model output frequency. Return to the *Model Control* form,

1. Expand the *Timing/Linkage* tab.
2. RMC on the *Linkages* sub-tab to set the output frequency.

In the *EFDC+ Model Linkages* window that pops up (Figure 5.30),

1. Select *Link EFDC+ Results to EFDC+ Explorer* in the *EFDC+ Explorer Linkage* tab.
2. Set the *Linkage Output Frequency* as 60 minute. This setting means that EFDC+ will record the hourly simulation results in the model output files (Note: The smaller the output frequency, the larger the output file size).
3. Click *OK* to close the form.
4. Save the model.

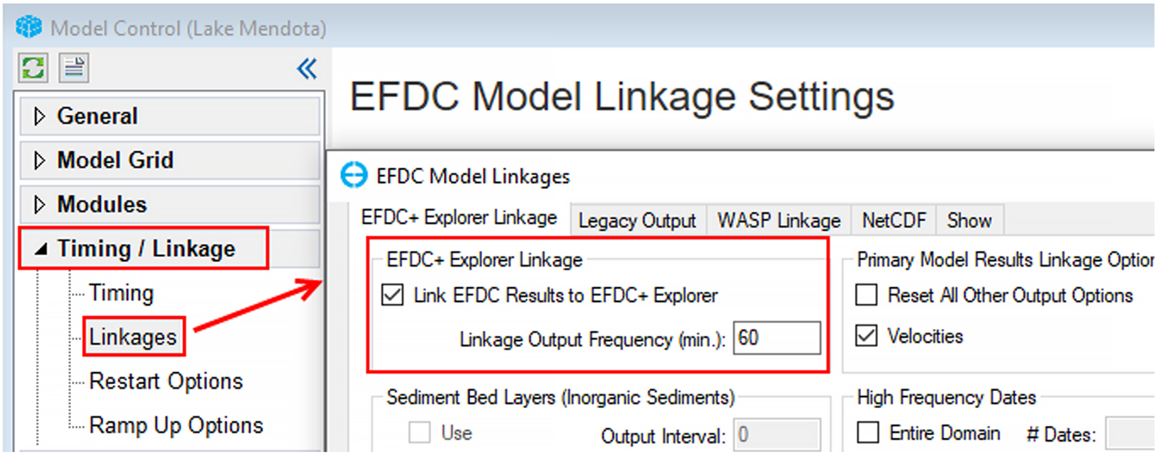

Figure 5.30: EFDC+ Explorer Linkage

5.7. Run Model

Now, we are ready to run the model. Follow the steps below to run the model (Figure 5.31).

1. Click the *Run EFDC+ button* in the main toolbar to prepare to run the model.
2. In the *EFDC+ Run Options* window popped up, click the *General* tab.
3. Set *OMP per Domain* to use for the EFDC+ run. Note that the *#OMP Cores Used* must be less than *Available CPU Cores*.
4. Browse to the executable file by clicking the (...) button (in a default installation, the executable is located in *C:\Program Files\DSI\EEMS11.6*)
5. Click the *Run EFDC+* button, then the model simulation will start.

Note: If the model has existing output files, a message line of "Overwrite the existing model outputs ?" will appear in the *EFDC+ Run Options* window. If so, please check the *Overwrite* box to run the model and save new results.

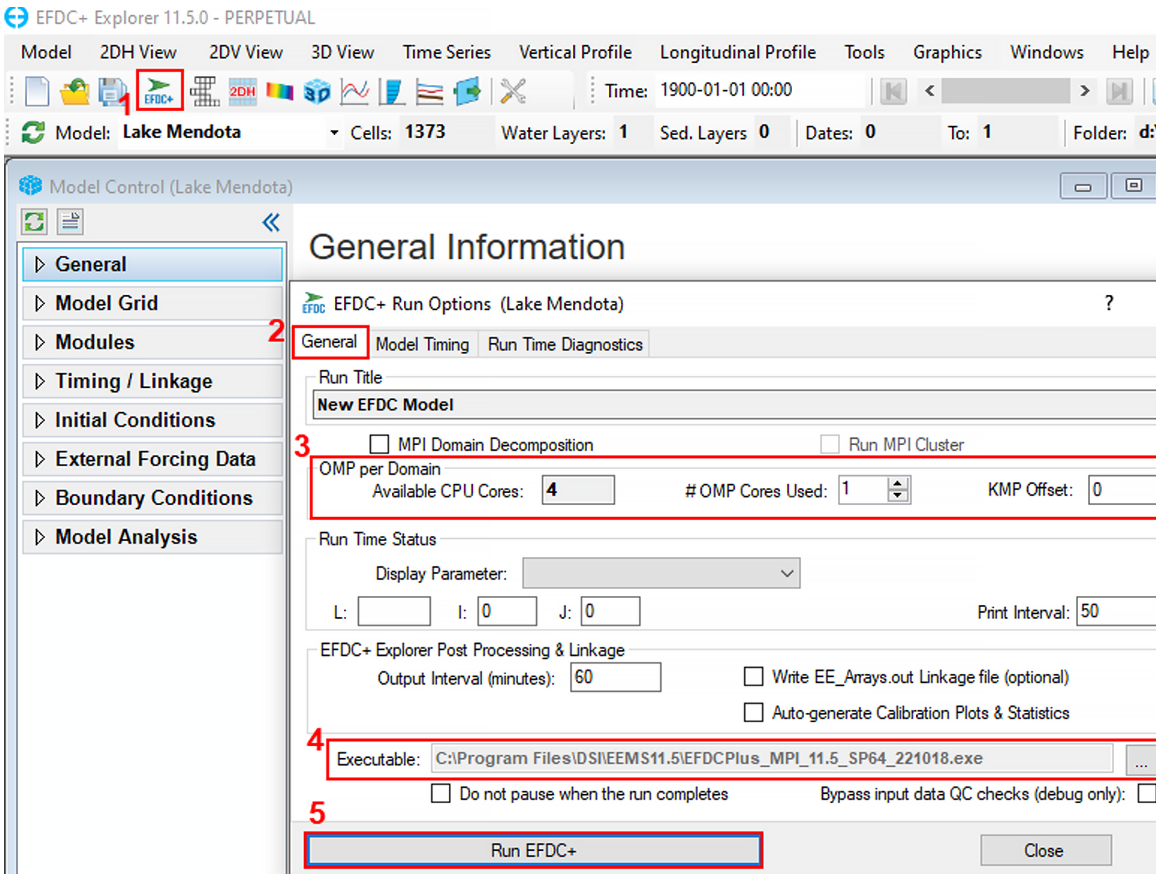

Figure 5.31: EFDC+ Run

5.8. View Model Output

This section provides the steps required for visualizing some model outputs visualization. After this exercise, you will be able to extract additional model outputs and visualize them using EE.

- After the model simulation is finished, click on the *Refresh Output* button in the toolbar to load the model outputs. You can do this step while the model is running as well. It will give you access to the partial model output.
- Click the *2DH View* button in the main toolbar .
- In the *Layer Control frame*, click on the light bulb symbol on *Online Background layer* (Figure 5.32) to display the online background map.

Note: To display the model grid for the correct online background map, the model grid needs to be set to the correct UTM Zone.

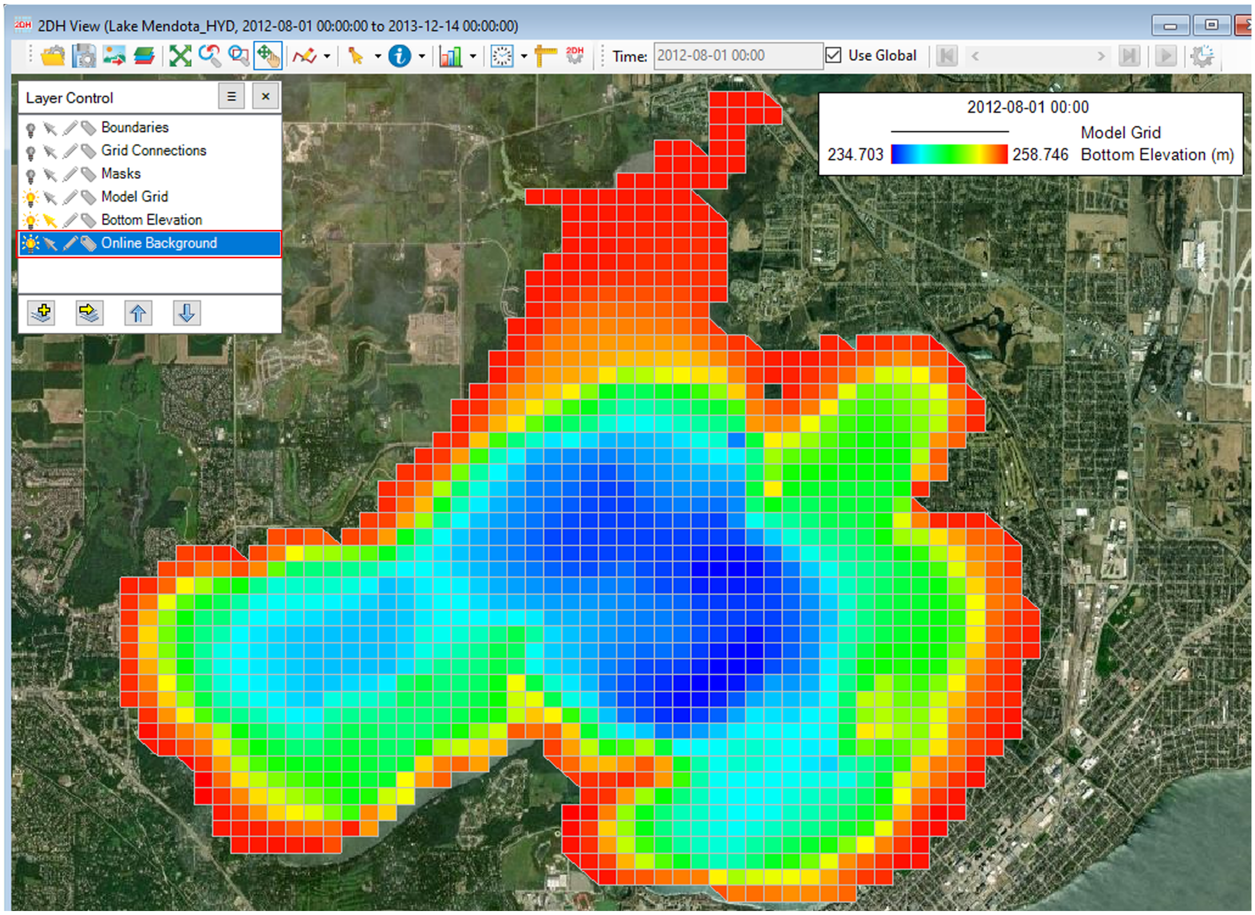

Figure 5.32: Online Background map for Lake Mendota

In the following steps, we will add Velocity Vectors to the 2DH map (Figure 5.33).

1. Click the *Add New Model Layer* button to open the *2DH View Option* form, which is used to select a layer to add to the *2DH View* map.
2. Select the desired parameter to add:
    - *Primary Group*: *Velocities*
    - *Parameter*: *Velocity Vector*
3. In the *Layer Setting* frame, choose *Depth Average* by default to make it simple.
4. Click on the *Add* button to add a layer to the *2DH View*.

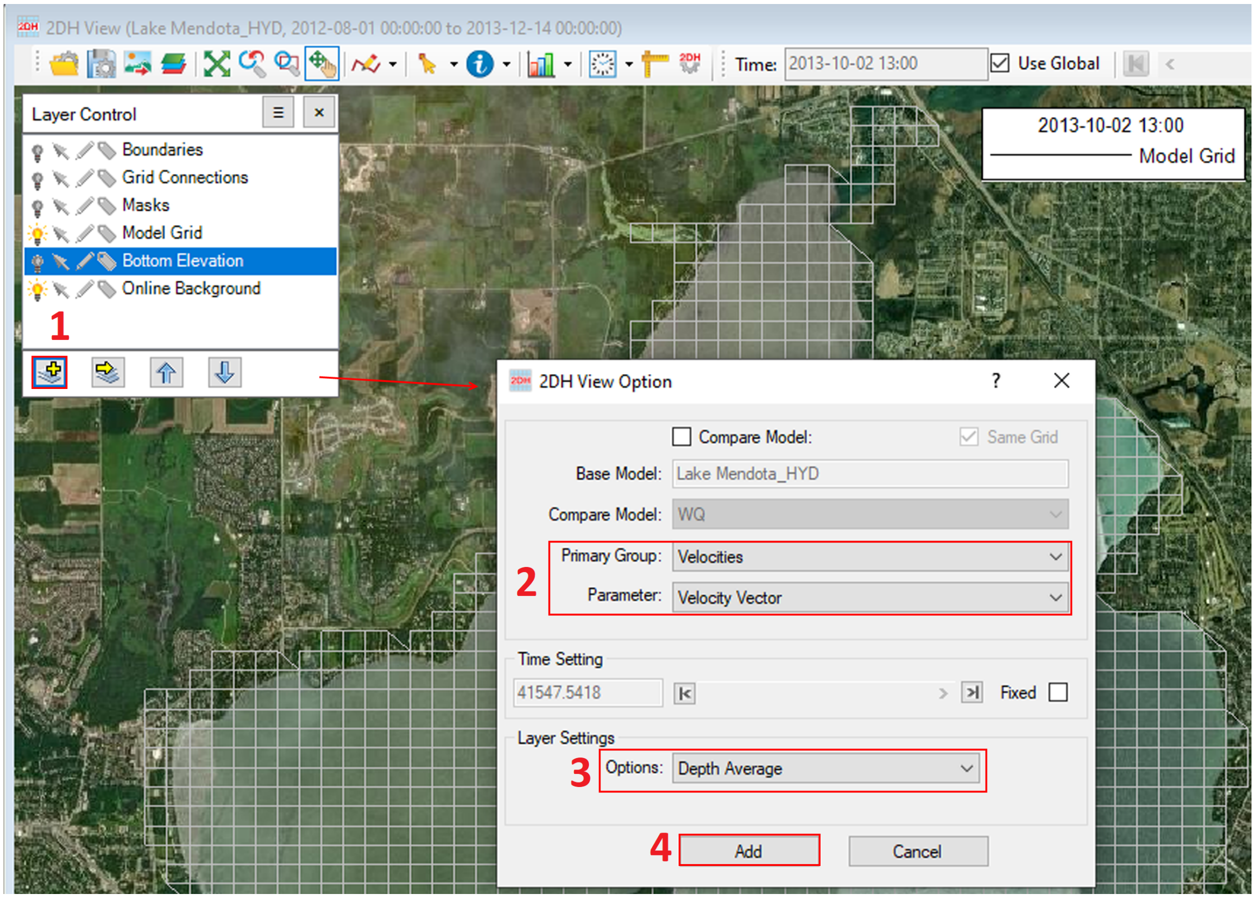

Figure 5.33: Add velocity vector layer in 2DH View