Exercise 4 – Introduction to WinHSPF and GenScn

BASINS/HSPF Application Steps

Study Definition/Modeling Strategy	Timeseries Data Management	Parameter Development/ Model Setup	Hydrologic Calibration/ Validation	Water Quality Calibration/ Validation	Scenario Analysis
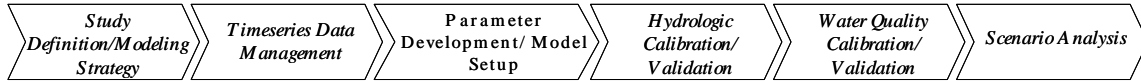
In this exercise, you will learn how to build and run a WinHSPF project. We will add three shape files (*outlets*, *streams*, and *subbasins*) to our BASINS project. These files are associated with the watershed delineation for the West Branch Patuxent River and are necessary to launch WinHSPF from BASINS. They can be created using the BASINS manual delineation tool or automatic delineation tool. Using these files, we will launch WinHSPF from the BASINS interface.

The WinHSPF interface includes many useful tools and user-input windows, many of which allow user-specified parameter adjustments. Because our goal during this exercise is to help you gain a basic understanding of WinHSPF and GenScn, we will leave most of the inputs as the default values. After we have explained each of the tools and some of the input tables, we will run our WinHSPF model for flow.

One of the most important aspects of modeling watersheds is hydrology calibration. In order to ascertain how well our model is simulating conditions in our watershed, we need to compare the model's output to observed data. In this exercise, we will download historical streamflow data, add it to our project's output file, and then use GenScn to plot the simulated streamflow on the same graph as the observed streamflow. In Exercises 5 and 6, we will learn how to calibrate our model's hydrology so that it more accurately predicts streamflow.

Questions addressed in this exercise:
1) How do I import *Subbasins*, *Streams*, and *Outlets* layers into my BASINS project?
2) How do I launch WinHSPF from BASINS?
3) What tools are available within WinHSPF?
4) How do I run my WinHSPF model?
5) How do I add gage data to my WinHSPF project file?
6) How do I use GenScn to view my simulation results?

## A. Importing delineation layers

QUESTION ANSWERED:
1) How do I import Subbasins, Streams, and Outlets layers into my BASINS project?

1.   If BASINS isn't open, from the **Start** menu under **Programs**, select **BASINS** and then **BASINS 4**.

2.   **Click** "*patuxent*" in the "Welcome to BASINS 4" window.

3.   To avoid confusion, remove the *Streams*, *Subbasins*, and *Outlets* layers produced during the previous exercise by right-clicking on each layer and selecting "Remove Layer". Also remove the results of the automatic delineation from the map legend.

              Note: Map layers representing *Streams* and *Subbasins* are required in order to launch WinHSPF from the BASINS interface. An *Outlets* layer is optional. These files can be created using the manual delineation tool or the automatic delineation tool. In this case these files were created by the automatic delineation tool. We will use these predefined map layers so that everybody will have the same model results.

4.   Click ⊞ (Add Layer tool).

5.   Navigate to *c:\basins\predefined delineations\west branch* and select all three shapefiles in the folder.
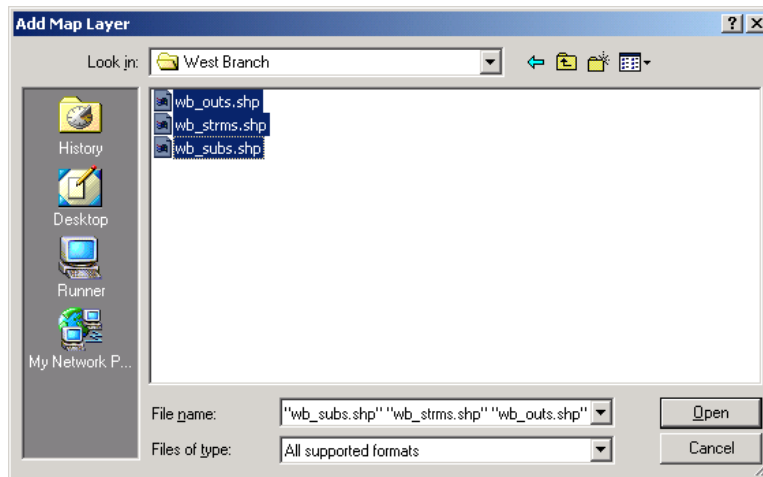
6.   Click OPEN. Three layers (Streams *wb_strms.shp*, Subbasins *wb_subs.shp*, and Outlets *wb_outs.shp*) will be added to the map.

7.   Turn "off" all layers except *Streams*, *Subbasins*, *Outlets, State Boundaries,* and *Cataloging Unit Boundaries*.

8.   Make sure the *Outlets* and *Streams* layers are displayed on top of the *Subbasins* layer. Your view should look like the following.
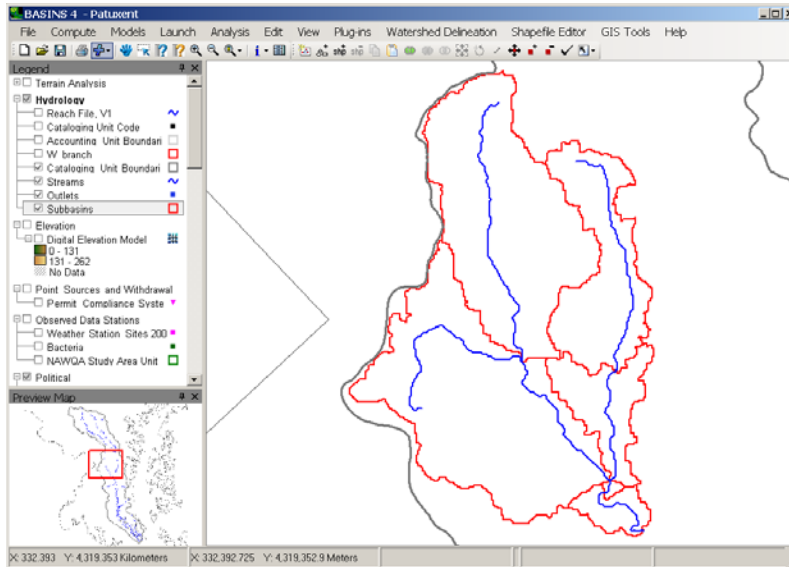
B. Launching WinHSPF from BASINS

QUESTION ANSWERED:
2) How do I launch WinHSPF from BASINS?

Now that we have added the new *Subbasins*, *Streams*, and *Outlets* layers, we will launch WinHSPF from the BASINS interface. When WinHSPF is launched from BASINS, an input file (*.uci*) is created. HSPF uses the information found in BASINS to estimate values for many parameter inputs. It should be noted that these values can be highly inaccurate and should be modified if more accurate information is available.

1.   From the **Plug-ins** menu, make sure **Model Setup (HSPF/AQUATOX)** is checked (on).
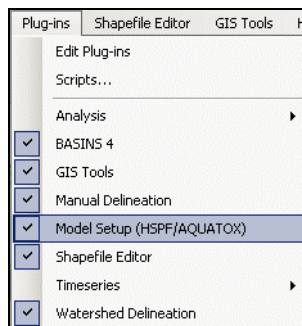
2.   Turn "on" and "activate" the *Subbasins* layer.

3.   Click (Select tool).

4.   While holding down the CTRL key, click on all 6 subwatersheds.
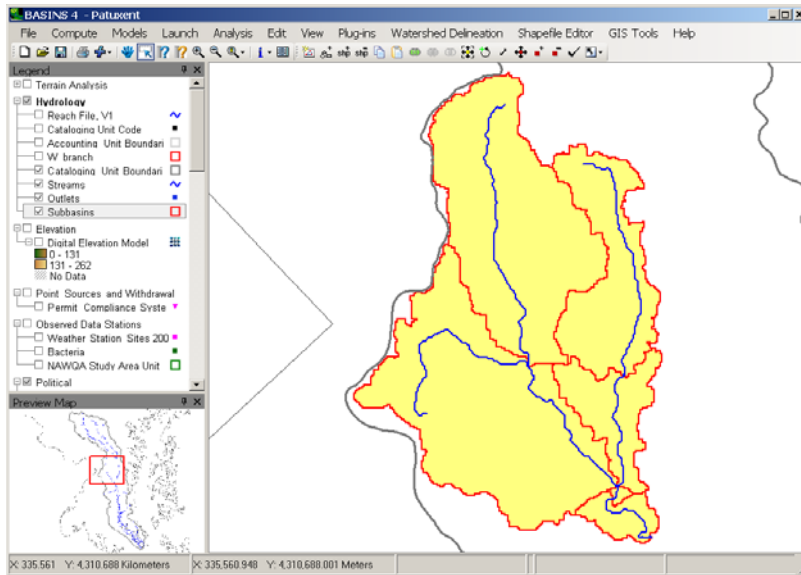
5.   From the **Models** menu, select **HSPF**. (Be patient for a moment.)

6.   The following window will appear. Enter "WestPat" in the "HSPF project name:" box. Leave "USGS GIRAS Shapefile" in the "Land Use Type" box. Set *Subasins* to be the "Subbasins Layer", *Streams* to be the "Streams Layer", and *Outlets* to be the "Point Sources Layer". *Weather Station Sites 2006* will default to the "Met Stations Layer".
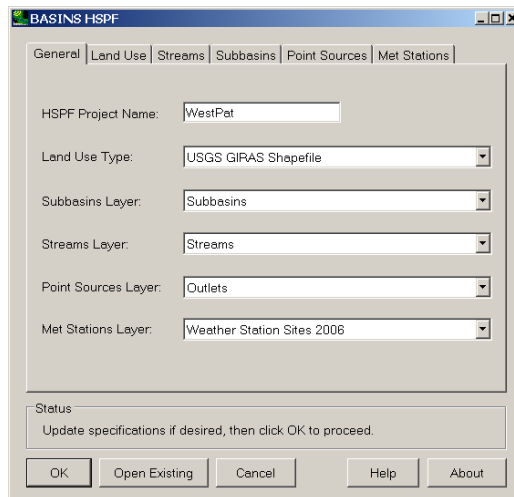
7.   Click the **Land Use** tab. The "Land Use" tab is used to specify the land use classification scheme. This scheme includes information regarding the combined land use classifications that will be used in HSPF as well as the impervious percentages for each of the land use classifications.
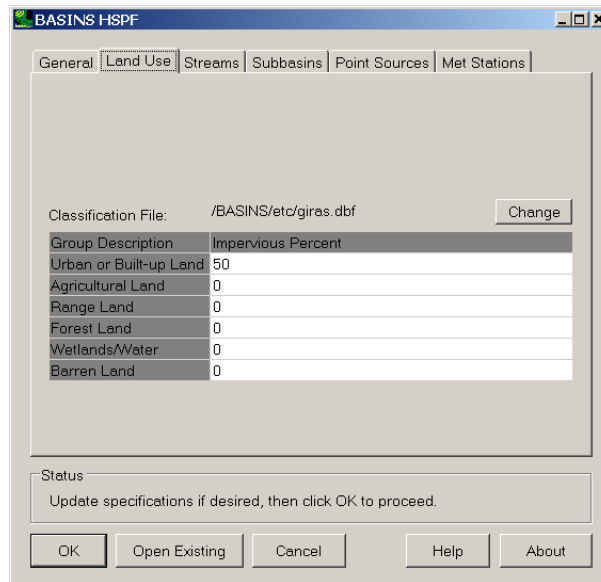
> **Note:** The "impervious percent" values indicate how much of each land use category
> will be impervious to infiltration, such as paved area and rooftops. For
> example, a 30% impervious percentage indicates that the given land use
> category is 30% impervious and 70% pervious. The model uses this
> information to estimate runoff from each land use type.

8. The Streams, Subbasins, and Point Sources tabs are used to map fields from those
   map layers to variables used in building the HSPF model. These fields will default
   appropriately when using the output of a BASINS watershed delineation tool, so we
   will not have to change anything in these tabs for this example.

9. Click on the **Met Stations** tab. Because we specified *Weather Station Sites 2006* as
   the "Met Stations Layer", the "Met WDM File" will be set to the WDM file
   associated with the *Weather Station Sites 2006*, which is named 'met.wdm' and is
   located in the '\BASINS\data\02060006' folder.

The list of the **Met Stations** tab will be populated with an identifier/description
corresponding to each meteorological station available in the WDM file for use in the new
HSPF project. One item from this list must be selected in order to have meteorological data
included in the HSPF simulation. (In Exercise 5 we will learn how to set up a new HSPF
project utilizing multiple met stations.) You should specify the meteorological station that
best represents your watershed.

The 'Select' button may be used to select another meteorological data file. Any WDM file
can be referenced as a met WDM file (as long as the file contains the necessary
meteorological data).
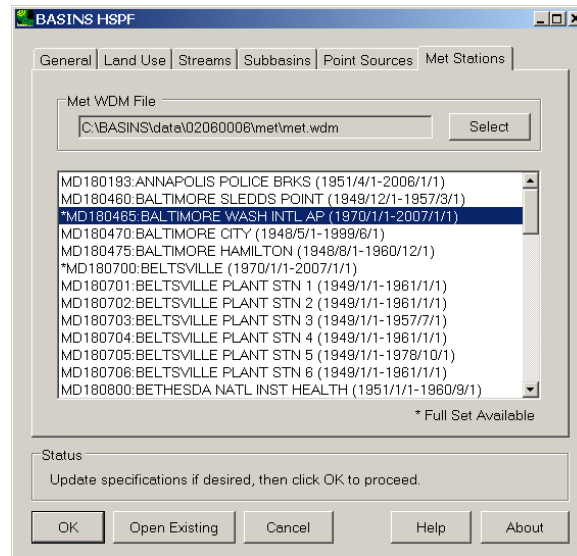
10.   Select "MD180465: BALT-WASHGTN INTL AP" as the Met Station to use in this HSPF project.

To determine which weather station to use, you may simply turn "on" and "activate" the *Weather Station Sites 2006* layer within BASINS, and then use the "Identify" tool to determine which station is closest (or most representative) of your watershed.

The "MD BELTSVILLE" station is actually closer to our watershed than the Baltimore-Washington Airport station is; however, the Beltsville station that is included in the BASINS database has some precipitation data gaps and therefore won't be used for this particular computer session.

The '*' next to the met station name indicates that this station has the full set of meteorologic constituents available. Stations without the '*' can be used as the meteorologic inputs for an HSPF hydrology simulation, but some met constituents will have to be specified from other locations to simulate full range of HSPF water quality modules.

11.   Click OK.

        Note: You will see some processing messages appear along with a status bar at the bottom of the main BASINS window. During this time the HSPF User Control Input (UCI) file will be created for the specified project.

After clicking OK, BASINS will create the HSPF User Control Input (UCI) file as specified. The UCI file will be assigned the HSPF project name indicated on the **General** tab, in a subfolder of the same name within the *\BASINS\modelout* folder. For example, for this HSPF project a UCI file by the name *WestPat.uci* will be created in the *\BASINS\modelout\WestPat* folder. A new WDM file called WestPat.wdm will also be created in this folder to be used for storing the point source inputs to the model as well as any output time series from the HSPF simulation.

Once the UCI has been created, the main window of the HSPF interface (WinHSPF) will open. This window contains several menus, a tool bar, a vertical tab strip, a schematic of the watershed, and an auxiliary table. The vertical tab strip, watershed schematic, and auxiliary table will not be active until a project is active (until a *.uci* file is opened or a project is created). When WinHSPF is launched from BASINS, the corresponding *.uci* file will automatically be opened. The information in the tab strip, the auxiliary tables, and the watershed schematic are relevant to the open project.
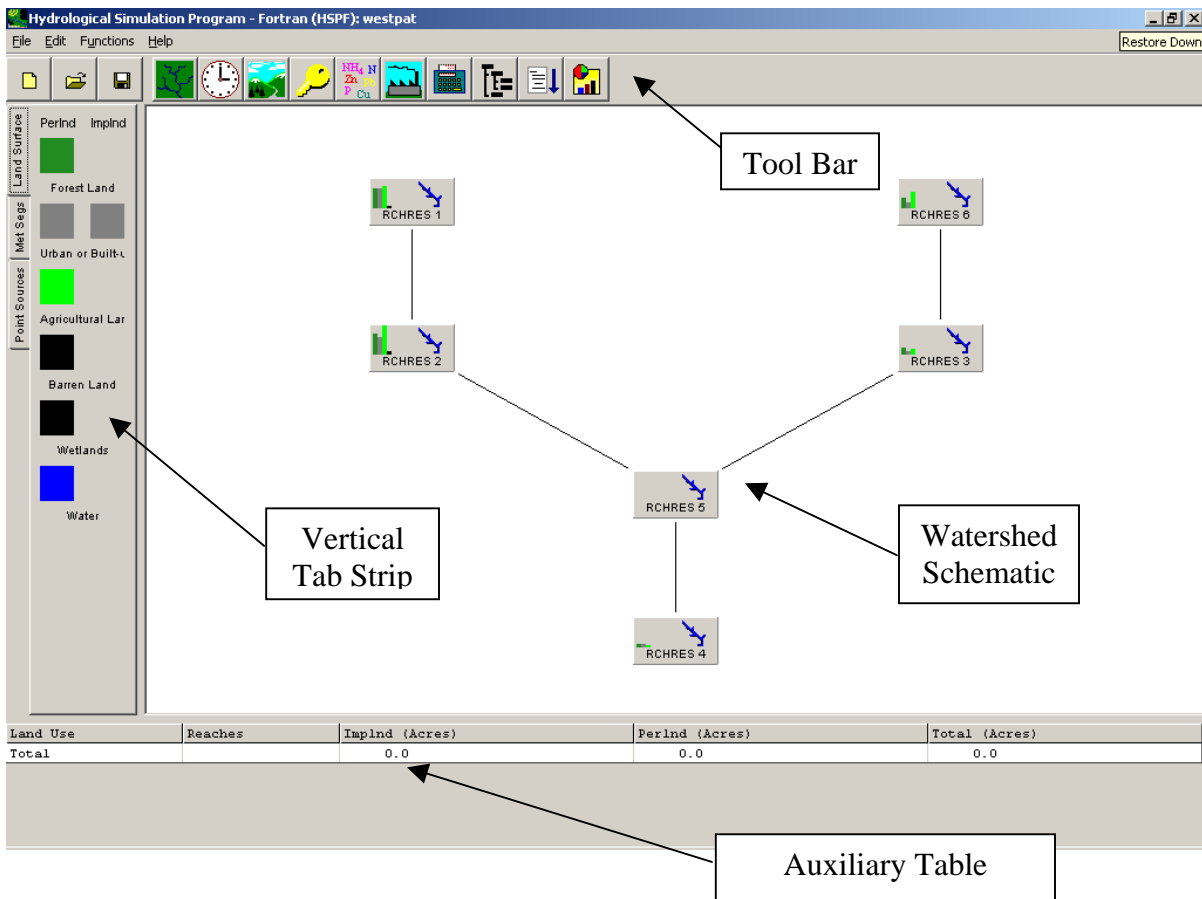
C. BASINS WinHSPF Tools

QUESTION ANSWERED:
4) What tools are available within WinHSPF?

The buttons on the tool bar, , provide links to the various tools with which the user can interact with the open project. The buttons to the left of the tool bar can be used for creating, opening, and saving projects. The "Run HSPF" button, , is used to perform the simulation (run HSPF). The other buttons on the tool bar are used to view and modify input values and parameters of the open project and to specify things such as the simulation period and constituents you would like modeled.

The vertical tab strip contains three tabs. The tabs, in conjunction with the figures in the

watershed schematic, display information about the project in the auxiliary table. These features can be used to specify whether to display information related to land surfaces, met segments, and point sources. If you highlight items in the tab strip and highlight a figure in the watershed schematic, related data will appear in the auxiliary table. For example, click the "Land Surface" tab and then click the "Forest Land" icon. Click the "RCHRES 2" figure in the watershed schematic. The auxiliary table will show the acres of forested area contributing to Reach 2.
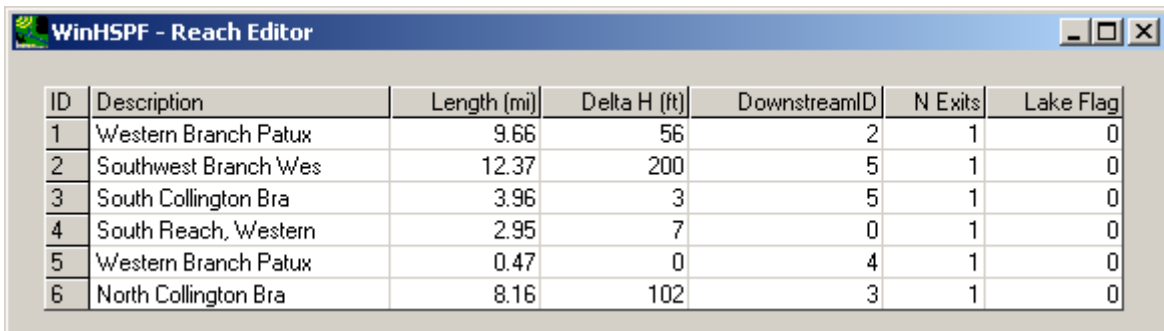
Clicking the "Met Segs" tab indicates which met segments are associated with each reach. The auxiliary table will show information for the selected met segment contributing to the selected reach. Information in the auxiliary table includes the constituent name, WDM data set number, associated multiplication factor for the PERLND/IMPLND segments, associated multiplication factor for the RCHRES segments, and the transformation function used to adjust the time dimension of the data units as necessary (Duda et al., 2001).

The "Point Sources" tab is used to view the point sources that will contribute to the streams in WinHSPF. If you click on a point source, the stream segment to which it will contribute flow and/or constituents will become highlighted in the watershed schematic and data regarding the source will appear in the auxiliary table.

In order to become more familiar with the WinHSPF interface, we will click on each button in tool bar and view the various tools, options, and default input values.

WinHSPF Reach Editor

1. Click (Reach Properties Editor). This editor allows you to modify select properties of each reach. The following screen will appear:
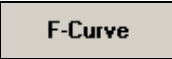
| ID	Description	Length (mi)	Delta H (ft)	DownstreamID	N Exits	Lake Flag
1	Western Branch Patux	9.66	56	2	1	0
2	Southwest Branch Wes	12.37	200	5	1	0
3	South Collington Bra	3.96	3	5	1	0
4	South Reach, Western	2.95	7	0	1	0
5	Western Branch Patux	0.47	0	4	1	0
6	North Collington Bra	8.16	102	3	1	0

> **Note:** The values in the "Reach Editor" table are automatically calculated when either the manual or automatic delineation tools are used to create the *Streams* layer. If neither of these tools is used, the user must make sure these fields and their corresponding values are included in the attributes of the *Streams* shapefile before launching WinHSPF.

2. Click FTABLES. The "Edit Ftable" window will open.
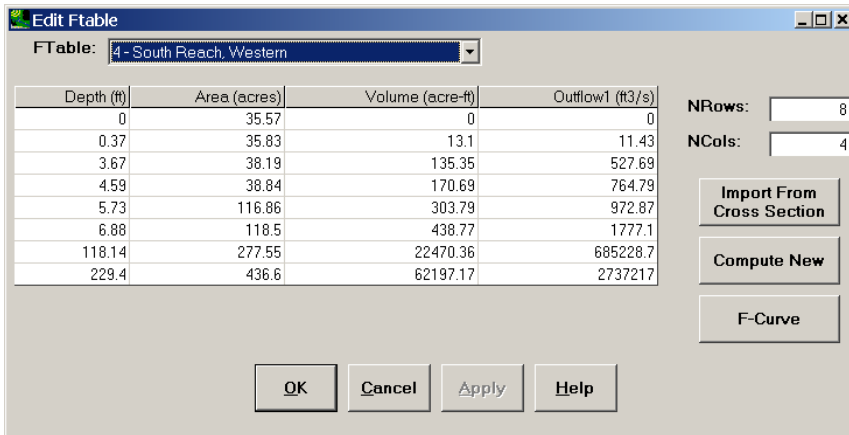
> **Note**: WinHSPF estimates the depth and width of individual reaches using some simple assumptions and the drainage area of a subbasin. These estimates are generally inaccurate, but they are better than nothing. It is best to gather data for your watershed's individual reaches and then use cross-sectional characteristics to populate the F-Tables by either estimating the stage-discharge relationship with an equation such as the Manning's Equation and then entering them manually or by using the "Import From Cross Section" tool. In this exercise, we will use the default F-tables.

3.   Click [**F-Curve**]. A graph that shows the relationship between the depth (stage) and reach flow rates will appear.

4.   From the **File** menu, select **Close**.
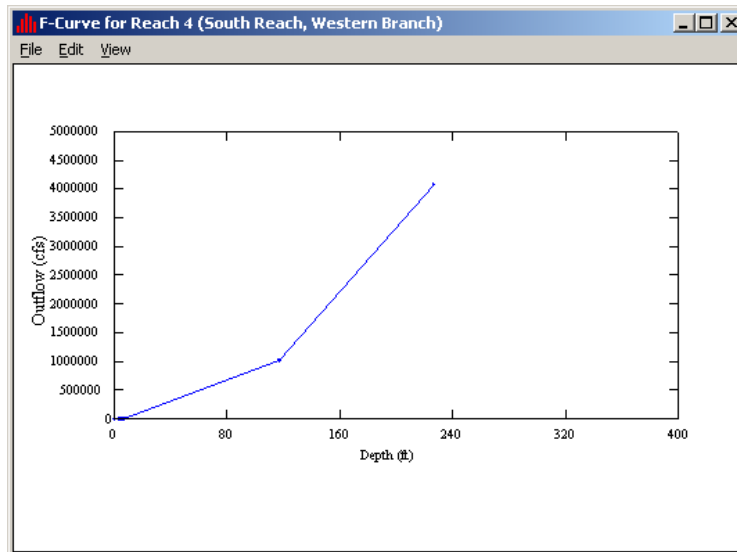
5.   Click [**Import From Cross Section**].
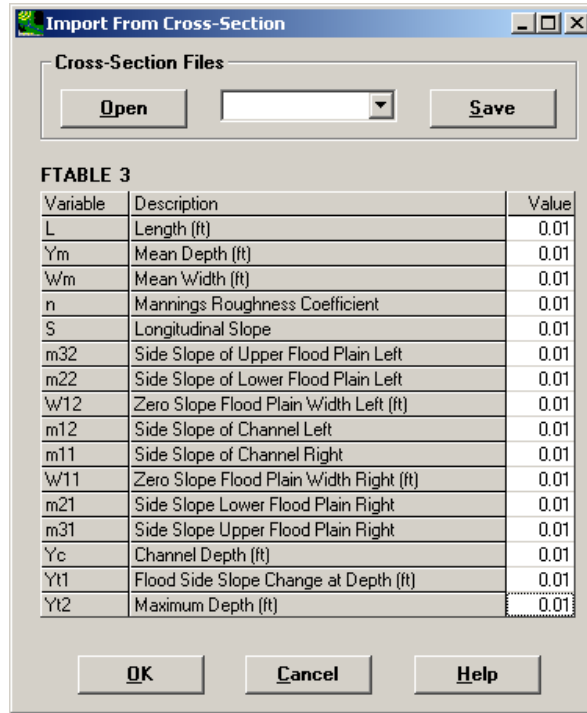
> **Note:** Using this editor, you can modify the cross section characteristics or import a table that contains the cross-sectional data (*.*rtf* file).

6.   Click CANCEL to exit the "Import From Cross-Section" window.

7.   Click OK to exit the "Edit FTable" screen.

8.   Click OK to exit the "Reach Editor" window.

WinHSPF Simulation Time Editor

A *.*wdm* file containing meteorological time series data was specified in the "Create Project" window. In the Simulation Time Editor, you should specify a simulation period (the default time period is for all dates included in your meteorological data file – this time period is usually very long (several decades). The period you specify must be within the time range of data included in your meteorological *.*wdm* file.

1.   Click (Simulation Time and Meteorological Data).

2.   Enter the dates shown below for the 'Start' and 'End' of the simulation.
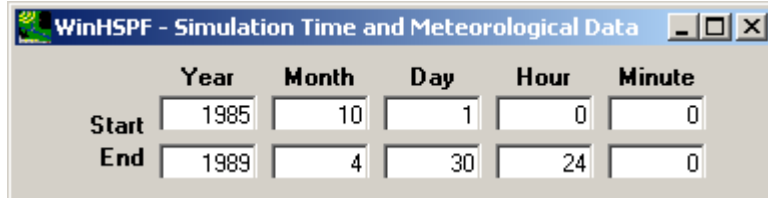
> **Note:** The Met Segments section of the Simulation Time and Meteorological Data window contains a grid with a list of the meteorological segments and the operations that they affect. New met segments may be added by clicking ADD button. Click EDIT to modify existing met segments. To edit a met segment, highlight the desired segment in the "Met Seg ID" column then click EDIT. The "Add Met Segment" and "Edit Met Segment" forms are almost identical, except that the Met Segment name in the 'Add Met Segment' form may be chosen from a drop-down list. The precipitation data set defines the met segment and therefore, may not be edited in the "Edit Met Segment" form (Duda et al., 2001). We will not add or edit any met segments at this time.

3.   Click OK.

WinHSPF Landuse Editor

This tool allows you to edit the area of each land use contributing to each reach and is useful for best management practices and modeling various scenarios. This will be further explained in Exercise 14.

1.   Click (Landuse Editor).
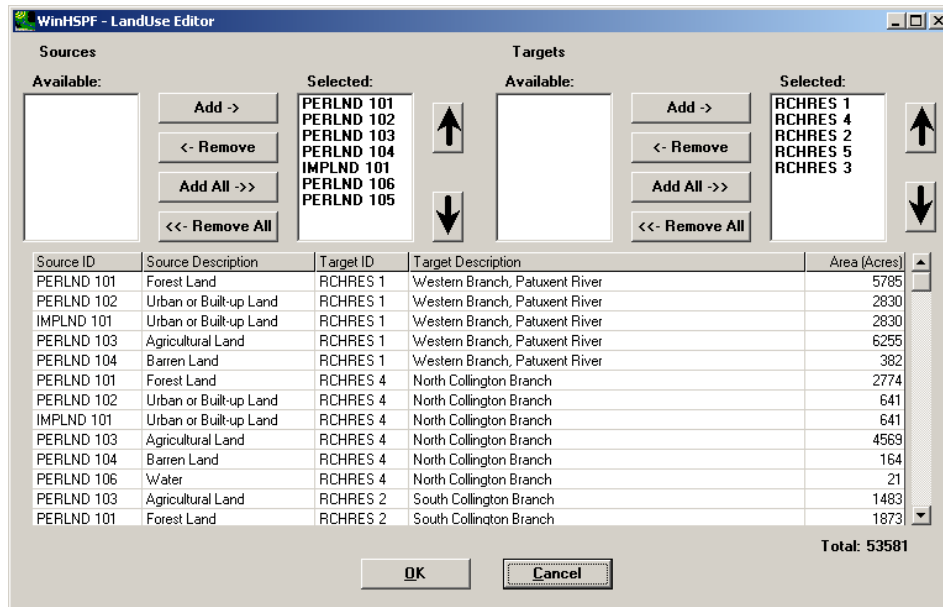
> **Note:** The "Sources - Available" box on the left represents the sources and land segments, and the "Targets - Available" box on the right represents the targets

and reaches. The table at the bottom of the window lists each "selected" land segment contributing to each "selected" reach. Reaches and land segments are represented in the table if the name of the operation is in the "Selected" box. Sources, land segments, and targets can be moved back and forth between the "Selected" and "Available" boxes. The contents of the grid will automatically be updated.

> **Note:** The values in the "Area" column may be edited.   The text below the grid will indicate the original total area, the new total area, and the difference between these two. This information allows the user to feel confident that the change was interpreted as desired (Duda et al., 2001).

2.   Click OK.

WinHSPF Control Cards Editor

1.   Click ![key icon] (Control Cards).

2.   Click DESCRIPTIONS when the following window appears.
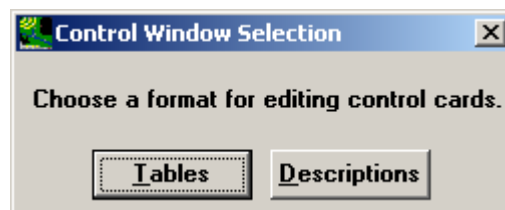
> **Note:** This screen allows you to choose which parameters you would like to model in WinHSPF. Each tab (PERLND, IMPLND, and RCHRES) contains a list of corresponding modules and parameters. You can specify which parameters and modules you want to model by checking the boxes beside the parameters. For example, if you wanted to model the accumulation and melting of snow and ice, you would check the box beside "Snow." When you turn "on" (check the box beside) a module through this editor, other sections might turn on automatically because there are rules in the model's coding that require certain modules as prerequisites to others. These operating rules are based on the guidelines in *BASINS Technical Note 3: the NPSM/HSPF Simulation Module Matrix*. Following these operating rules, all required and recommended prerequisite sections are turned on when appropriate (Duda et al., 2001).

3.   The "Pervious Land" tab will be displayed. Make sure that "PWATER" (water budget pervious) is selected.
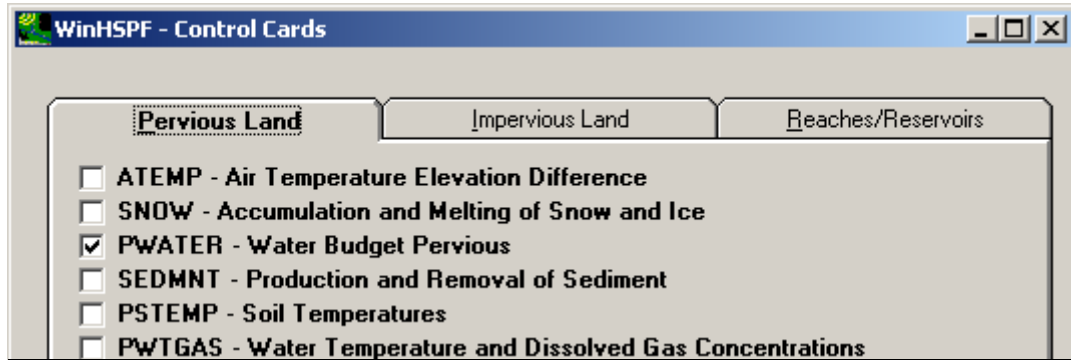
4.   Click the "Impervious Land" tab.
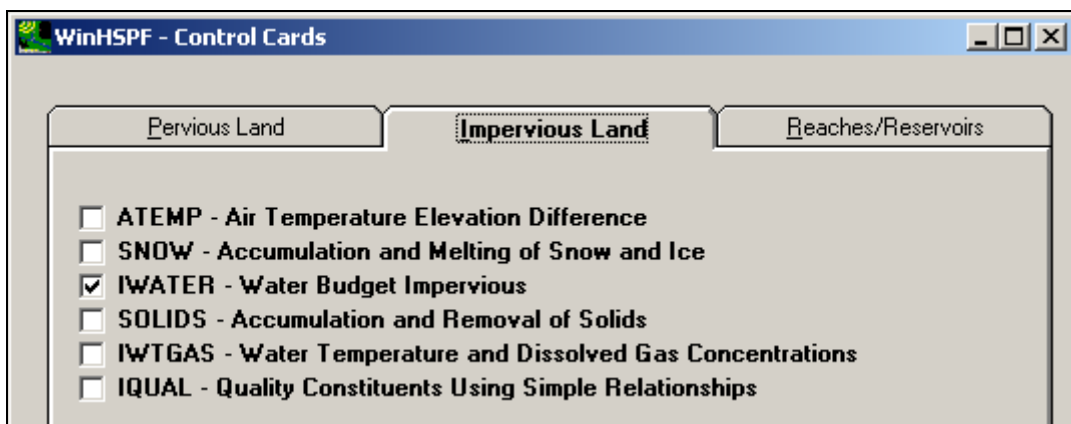
5.   Make sure that "IWATER" (water budget impervious) is selected.

6.   Click the "Reaches/Reservoirs" tab.
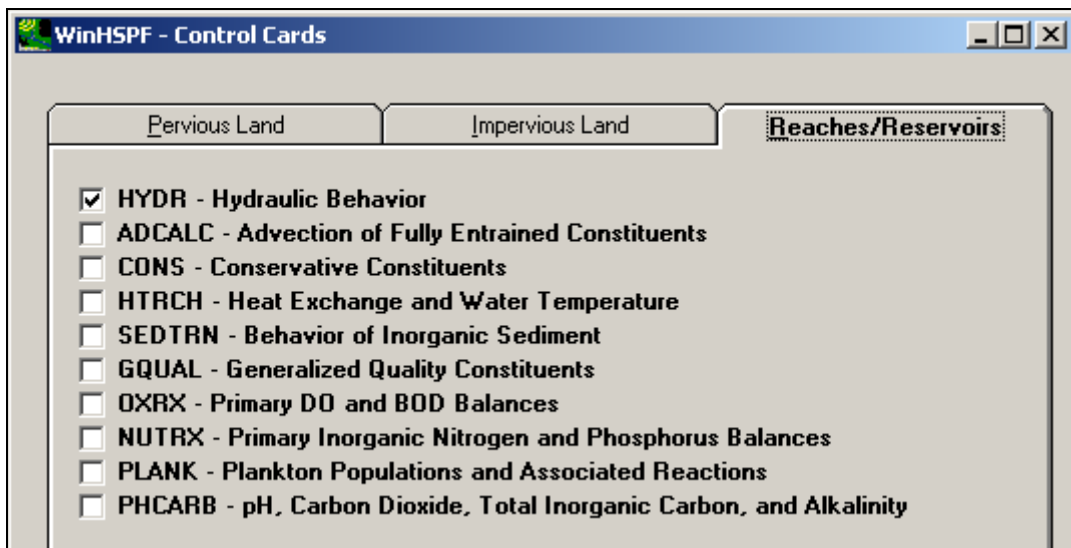
7.   Note that "HYDR" (hydraulic behavior) is selected, and select "ADCALC."

> **Note:** Notice that PWATER is selected in the Pervious Land tab, IWATER is selected in the Impervious Land tab, and HYDR is selected in the Reaches/Reservoir tab. These three selections indicate that we will be modeling overland flow in pervious and impervious land segments and it will then be routed into the stream reach.

8.   Click OK.

Pollutant Selection

1.   Click (Pollutant Selection). In this window, we can specify which water quality constituents we would like to model using HSPF. We will not change anything in this screen since we are not modeling water quality in this session.
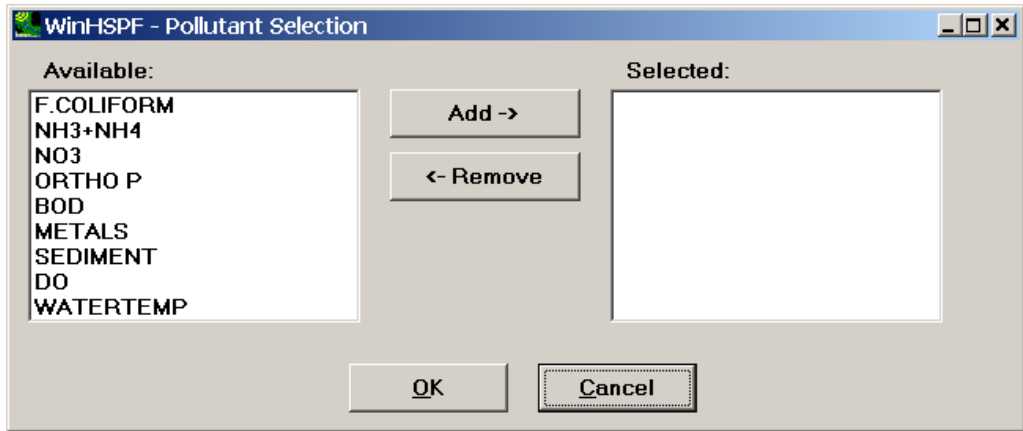
2.   Click OK.

Point Source Editor

This tool is used to view and manage point sources in the simulation. We will add the PCS point source that contributes to Reach 4.

1.   Click (Point Sources). The "Point Sources" window will open.
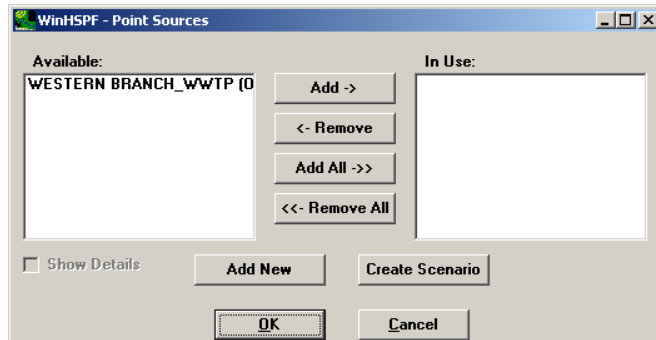
2.   In the "Available" box, select "WESTERN BRANCH WWTP (OBS)" and click "Add->." The record will now be in the "In Use" box.
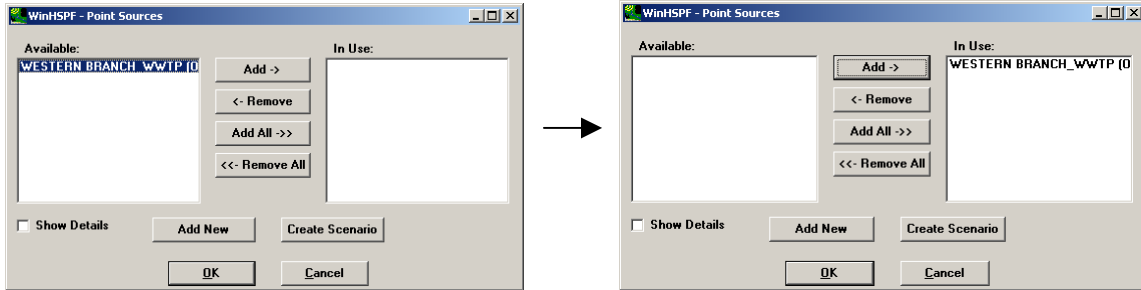
3.   Click in the box next to "Show Details." The Point Sources editor will expand.
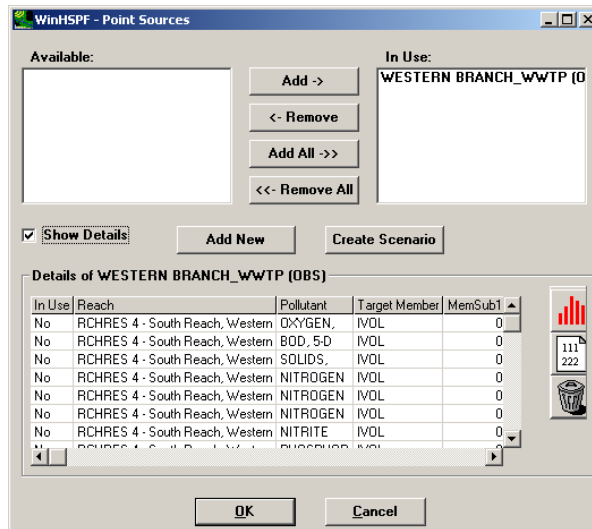
4.   The list under "Details of WESTERN BRANCH_WWTP(OBS)" shows the pollutants included in the PCS file. The model will not simulate a constituent unless its corresponding "In Use" box is set to "Yes." Scroll down until you see "FLOW" in the "Pollutant" column. Double click the corresponding "In Use" box and select "Yes."
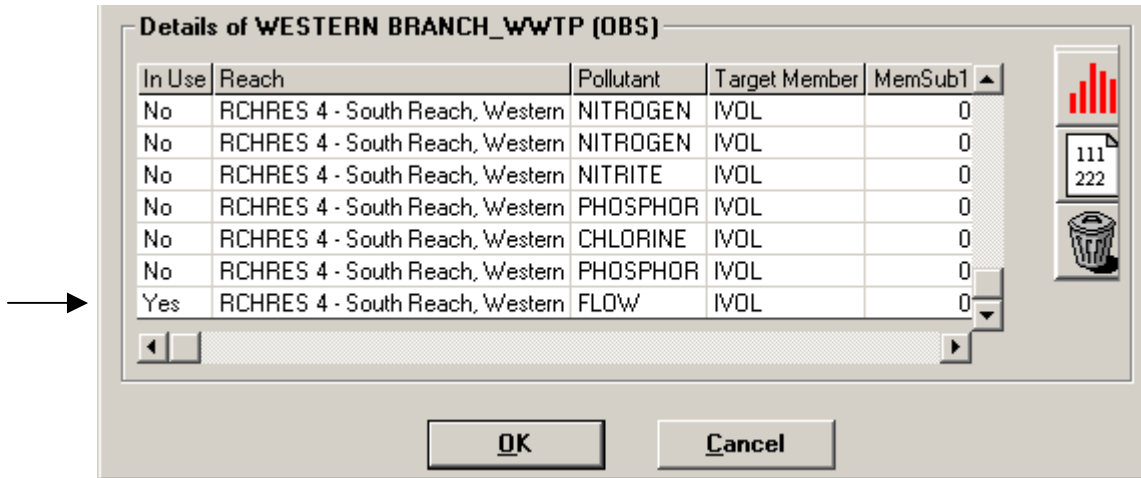
5. Click OK.

Note: If you want to use point source data other than that provided by BASINS, click ADD NEW. The following window will appear.
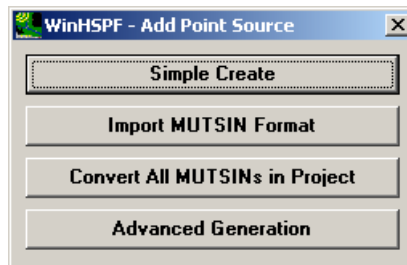
The following gives explanations for the buttons on the "Add Point Source" window.

Simple Create: This tool allows you to add point source data by specifying a scenario name, a reach to which to add the point source, the pollutant(s), a facility name, and a daily load.
Import MUTSIN Format: This allows you to select a Mutsin file that contains the point source information. The user must specify a scenario name, a reach to which to add the point source, the pollutant(s), and a facility name.
Convert All MUTSINs in Project: This tool scans the HSPF input sequence (the UCI file) and converts any found Mutsin files to WDM data sets in the project WDM file.
Advanced Operation: This tool allows you to modify the time series in your point source data file.

Note: You might want to edit values in one of the point source data sets. You can do so by highlighting a field in th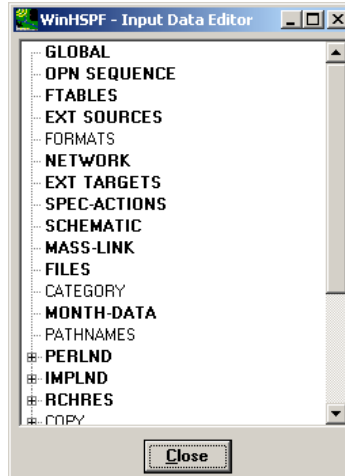e row associated with the desired data set and then clicking the "List/Edit Selected Point Sources" button, . Also, you can view the results graphically by clicking the "Graph Selected Point Sources" button, . You can delete a point source by clicking the "Delete Selected Point Sources" button, .

WinHSPF Data Editor

1.   Click (Data Editor). The following window will appear.

> **Note:** The "Input Data Editor" window contains a tree diagram of all the possible blocks and tables of HSPF. The tree diagram is navigated by clicking on an item name. **Double clicking** on an item name reveals the sub-items within that item if any exist.
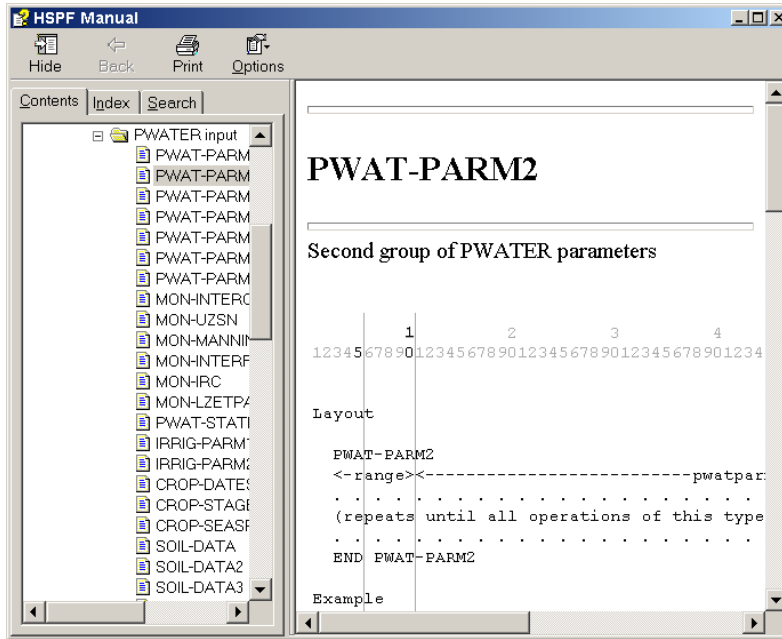
2.   Click ⊞ next to "PERLND."

3.   Click ⊞ next to "PWATER."

4.   **Double click** "PWAT-PARM2."
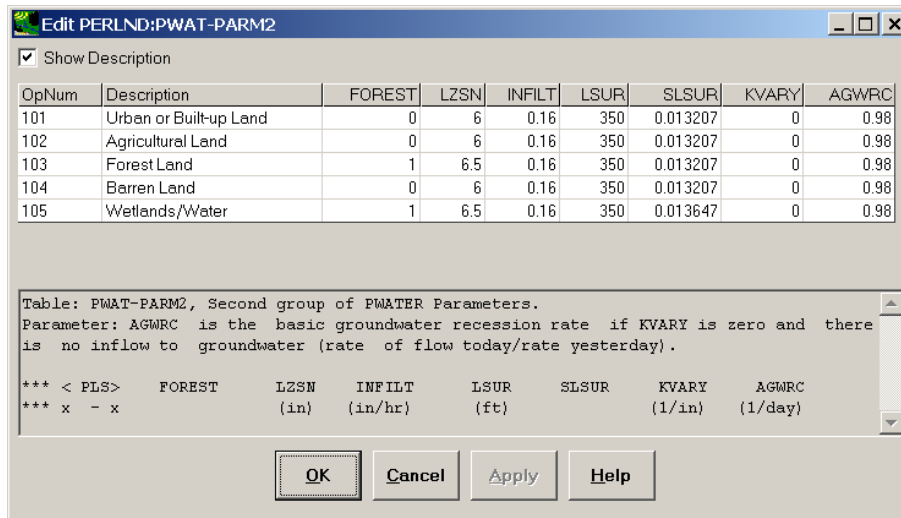
> **Note:** This editor is used to modify site-specific and calibration parameters varied in

model calibration. Notice that as you click in each column corresponding with a parameter, the definition of that parameter is displayed in the text box below the table.

5.   Click **HELP**.

Note: Notice that the HSPF Manual opens to the page corresponding to the table PWAT-PARM2. This page of the manual provides additional details about the individual parameters in the PWAT-PARM2 table, including definitions and limits.

6.   Scroll up to the top of the "Contents" at the left side of the manual.
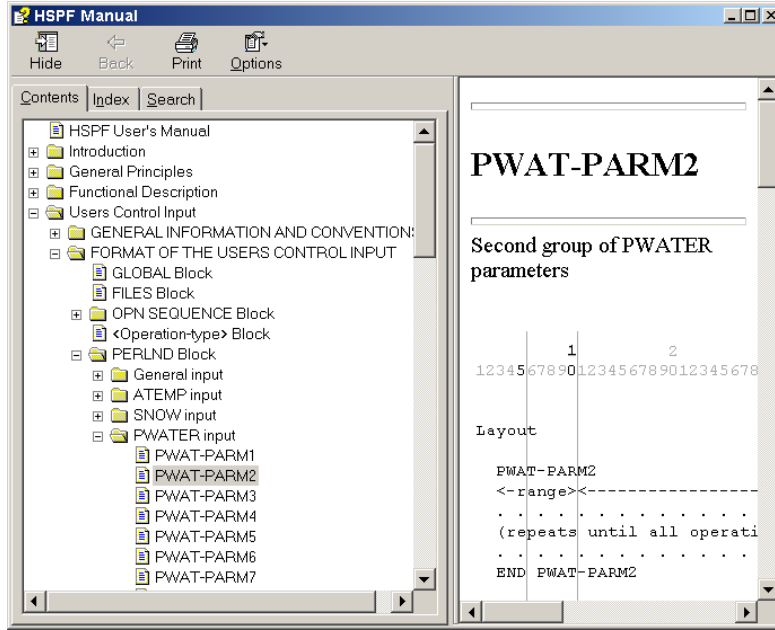
Note: The tree diagram in the "Contents" shows the general format of the HSPF manual. The "Functional Description" section contains detailed explanations of the algorithms and scientific basis of HSPF, while the "Users Control Input" section contains operational details.

7.   Scroll down in the "Contents." Within the "User Control Input" section, open the 'Catalog for PERLND module' folder, and click on the 'Group PWATER' page.
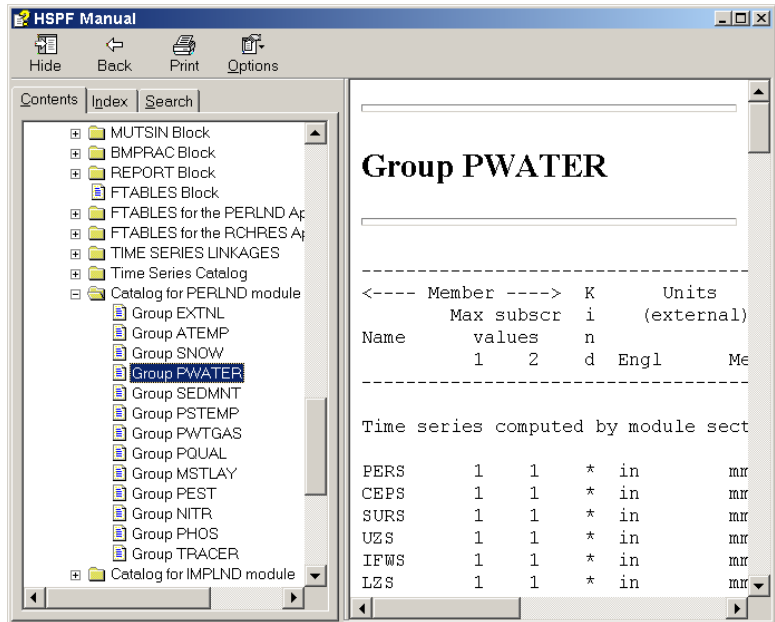
Note: This page is known as the "Time Series Catalog" for PWATER. The "Time Series Catalog" lists all time series computed by each HSPF module, including the units of the output. It also lists the possible input time series for each

module. The list of input time series indicates either prerequisite sections that must be turned on or time series that must be supplied externally (such as through the input WDM file) for a particular module section.

8.   Browse the HSPF Manual as much as you wish. When you are finished, close the manual by clicking the '**X**' in the top right corner.

9.   Toggle to WinHSPF. Click OK or CANCEL in the "Edit PERLND:PWAT-PARM2" window.

10.   Click **CLOSE**.

WinHSPF Output Manager

1.   Click (Output Manager). The Output Manager allows you to select what will be written as the output time series. For each output type, you can choose one or more locations for which output will be generated.

   Note: The first output type is Hydrology Calibration. This button will already be selected when entering the Output Manager. With this button selected, the list below the radio buttons displays the HSPF calibration locations within the current HSPF project. Under the buttons is a box explaining which output time series will be generated during the HSPF model run.

   You can add calibration locations to this list by clicking ADD. A window will appear that lists all available calibration locations (i.e., the reaches of the watershed) along with two text fields. The user must choose one of the calibration locations, and then enter an eight-character identifier for that location. This identifier is used as the location ID attribute in the WDM time-series data sets that will be created. The user may also enter a "base data set" number for the data sets to be created. The new data sets will be numbered consecutively as the available data sets following the base number. In this section, we are modeling only flow, so we will only specify output locations for "Flow."

2.   Click the radio button beside "Flow."
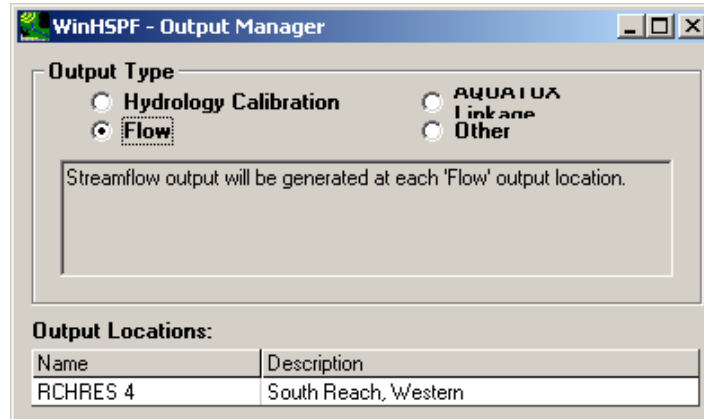
> **Note:** Notice that "RCHRES 4" is the only stream listed in the output locations. We would like to look at the flow in "RCHRES 5" because there is a flow gage located on Reach 5.

3.   Highlight "South Reach, Western" in the "Description" text box and click REMOVE.

4.   Click YES when the following message appears.
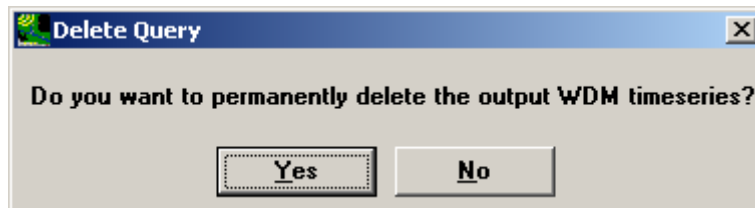
5.   Click ADD.

6.   Highlight "RCHRES 5 (Western Branch Patux)."
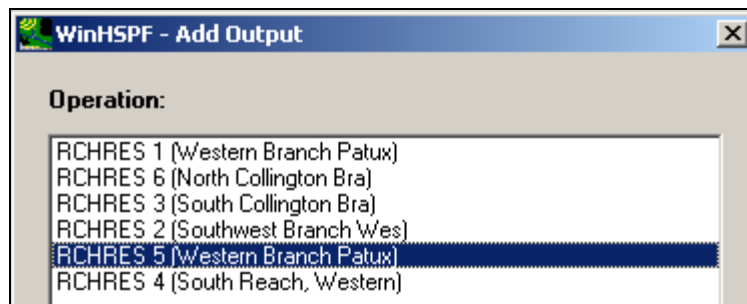
7.   Click OK. Notice that "RCHRES 5" is now listed as an output location. This means that we will be able to view the simulated flow data for Reach 5.

8.   Click CLOSE to exit the "Output Manager."

D. Running WinHSPF

In this section, we will run our WinHSPF model for the West Branch of the Patuxent River.

QUESTION ANSWERED:
4) How do I run my WinHSPF model?

1.   Click ⬛↓ (Run).

2.   If the following message appears and asks if you want to save changes that have been made, click **SAVE/RUN**. Your *.uci* file will be saved and the model simulation run will begin.

      Note: A screen will appear, indicating that the input file is being processed and run. When this disappears, the model run is complete.

      Note: We would like to compare the simulated flow for Reach 5 (recall that we specified RCHRES 5 as a "Flow" output location) with observed flow in Reach 5. Before we do this, we need to add the gage data to the current project.

3.   Close the WinHSPF window.

      IMPORTANT NOTE: IF YOU DO NOT CLOSE THE WINHSPF WINDOW, YOU WILL NOT BE ABLE TO COMPLETE SECTION E.

E. Adding Gage Data to the Project

QUESTION ANSWERED:
*5) How do I add gage data to my WinHSPF project file?*

In this section, we will download the USGS gage data for Reach 5 and add it to the *WestPat* project **.wdm* file. If you are unable to connect to the internet, please skip to step 13.

1.   Within BASINS, from the **File** menu, select **Download Data**.

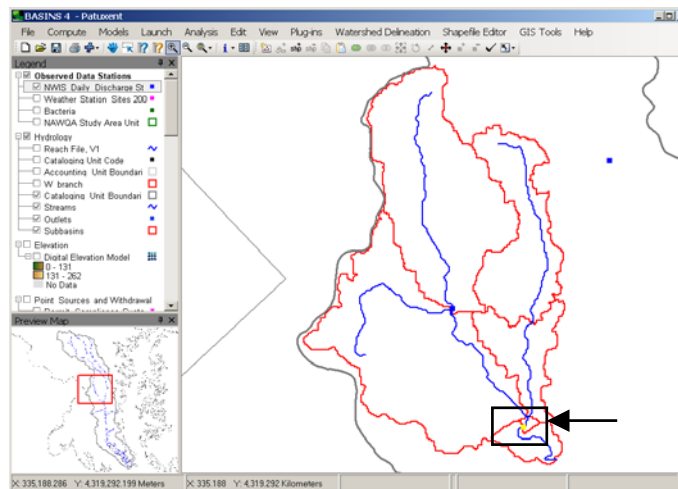2.   Check the box next to "Daily Discharge" within the "USGS Stations from NWIS"

frame. Your window should look like the following.
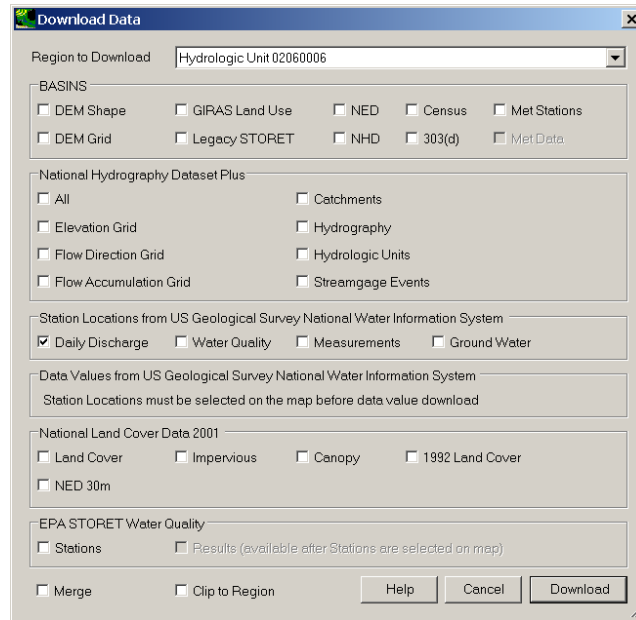
3.   Click DOWNLOAD. After a moment, the following window will appear.
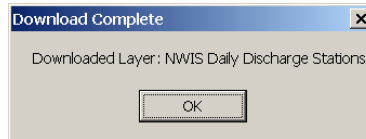

4.   On the map, notice the new layer called "NWIS Daily Discharge Stations". With this layer active, select the USGS daily discharge site at the downstream end of RCH 5, as shown on the map below. Note that you may want to move the Observed Data Stations to the top of the legend and change the point size/color to make the stations more visible.

The selected station is USGS discharge site "01594526" with a station name of "Western Branch at Upper Marlboro, MD". You could use the identify tool to be sure that you have selected the desired station.

Now that we have identified the station for which we would like to download the daily discharge data, we will use the Data Download tool to download the actual discharge data.

5.   From the **File** menu, select **Download Data**.

6.   Check the box next to "Daily Discharge" within the "USGS Data from NWIS" frame. Your window should look like the following.
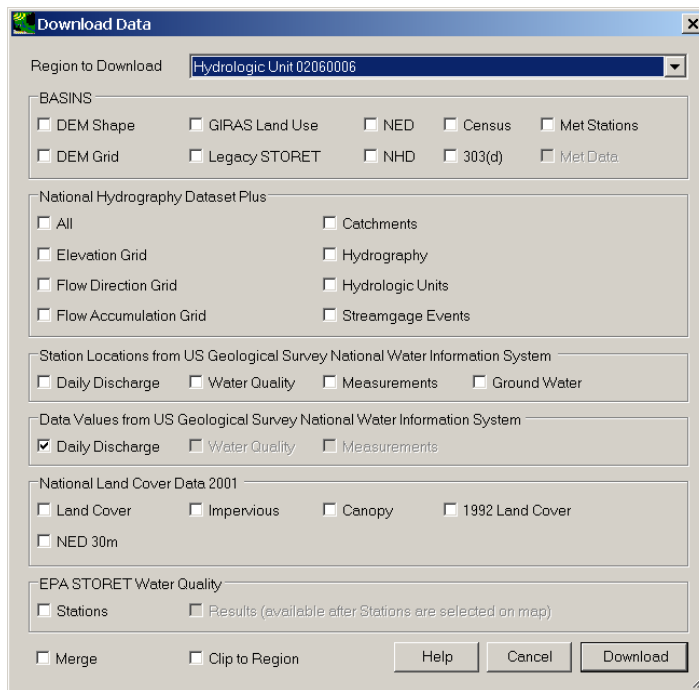
7.   Click DOWNLOAD. The following window will appear. Click 'Add data to existing WDM file', and then click Browse to select the file shown below:
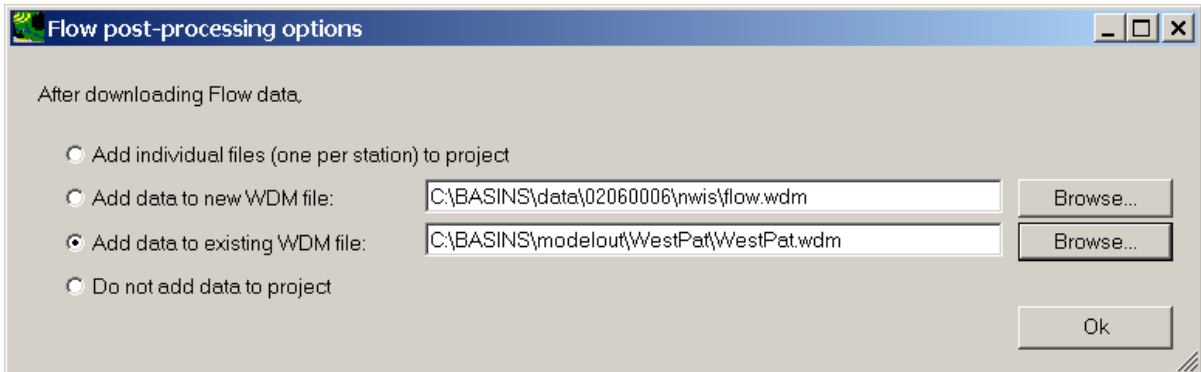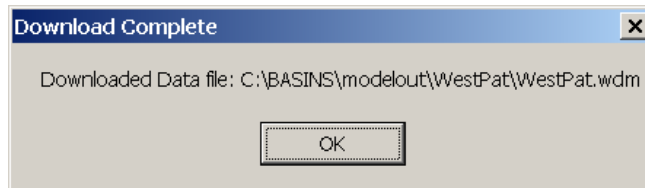
Note: In order to compare the simulated data to the observed data, we need to add the gage data to a WDM file. We will add the gage data to the project WDM file (*WestPat*) that was created when we built our HSPF project.
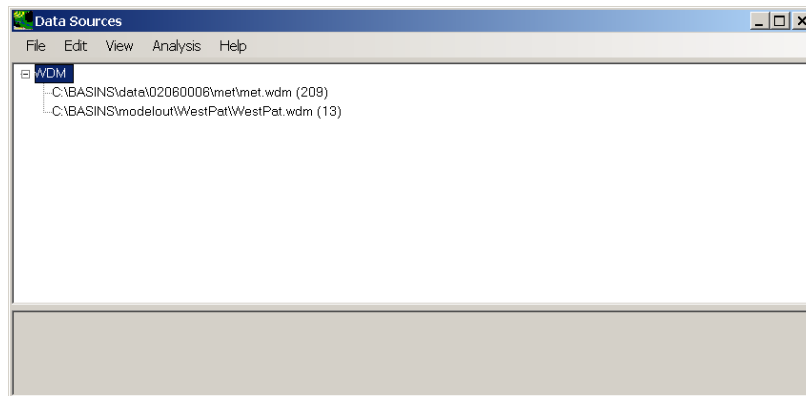
8.   Click OK. A message will indicate that 1 WDM data set was added.
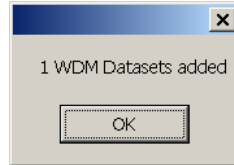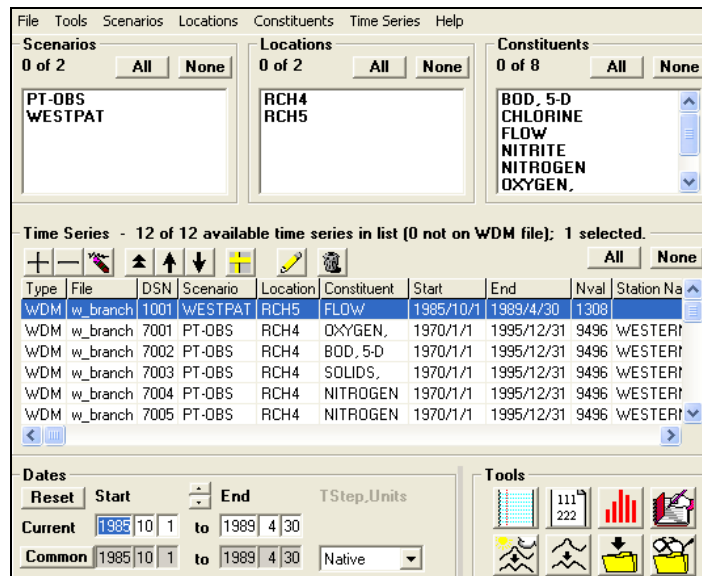

9.   Click OK.

10.   Click OK again.

11.   The following window will appear.

12.   Close this window by clicking the 'X' in the top right corner.

If for some reason you were unable to download the data through the **Data Download** tool, the downloaded data has been provided for you. You may import this data into the *w_branch.wdm* file using WDMUtil by going through the following steps. **If you were able to import the data successfully, you may skip steps 13 through 30 below and continue with Section F.**

13. From the **Start** menu under **Programs**, select **BASINS** and then **WDMUtil**.

14. From the **File** menu, select **Open**.

15. Navigate to *c:\basins\modelout\WestPat* and click on "*WestPat.wdm.*"

16. Click OPEN. The WDM will open:

17. From the **File** menu, select **Import**.

18. Navigate to *c:\basins\extra training files* and select "*discharge.txt.*" Click OPEN.

19. Click OK if the following message box appears.
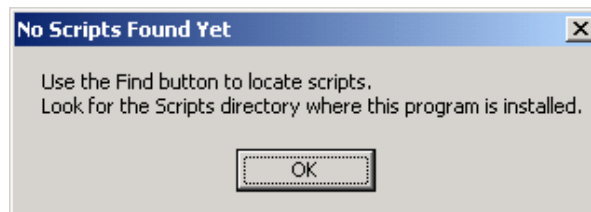
20. The "Script Selection" window will open. Click FIND.

21. Navigate to *c:\basins\models\hspf\WDMUtil\scripts* and select "*UsgsDvWeb_MDY_2001.WS.*"

22. Click OPEN.

23. Under script file, select the record that corresponds to "USGS Daily Web Values (mm/dd/yyyy) – 2001 Format." It will turn blue.
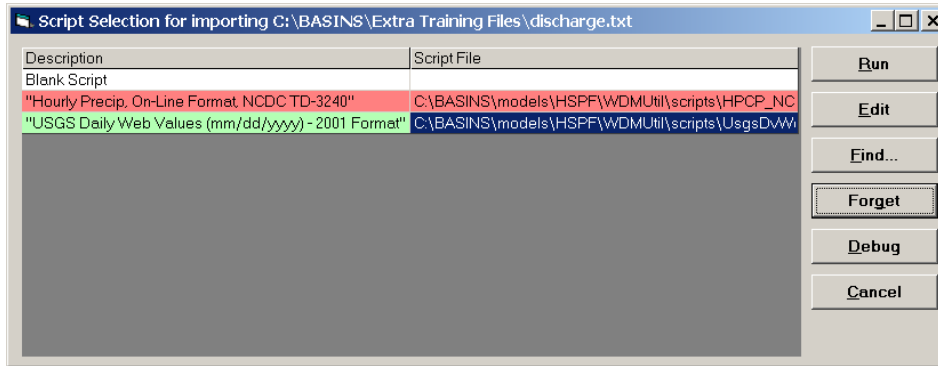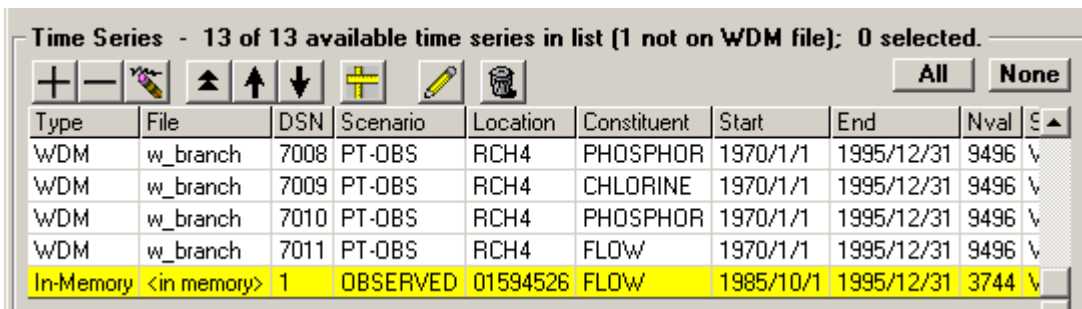
24.   Click RUN.

      The "Script Selection" window will disappear and a run window will open, indicating that the discharge data is being processed and added to the current WDM project.
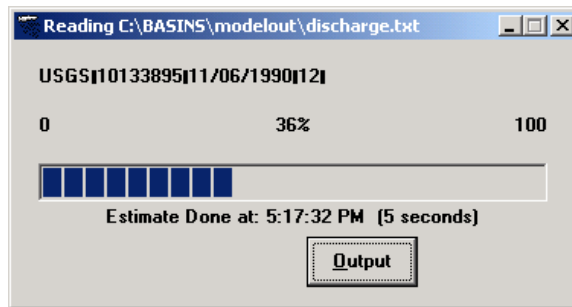
      When this process is complete, a new record will appear in the time series list. It will be highlighted (yellow).

25.   Click the new time series. It will turn blue when selected.

26.   In the "Tools" frame, click (Write Time Series to WDM).

27.   In the Output DSN text box, type "27." Leave other fields as they are.

```
Specify Output Data-set Number(s);
Select/Enter Scenario, Location, Constituent as needed;
Click Write button to store data on WDM file.
  ● Use common period for all data sets, as defined on main form
  ○ Use full period for each data set
```

DSN/ID	Output DSN	Scenario	Location	Constituent
1		OBSERVED	01594526	FLOW

```
                    [ Write ]      [ Cancel ]
```

28.   Click WRITE.

29.   Click OK when you are told that the file was stored successfully.

30.   From the **File** menu, select **Exit**.

F. Viewing WinHSPF Output in GenScn

QUESTION ANSWERED:
6) How do I use GenScn to view my simulation results?

In this section, we will view the results of the simulation using the GenScn postprocessor. GenScn was designed to manage large volumes of input, output, and calibration data used by and produced in models such as SWAT and WinHSPF. This software has many different applications; however, in this exercise we will focus on those necessary for viewing the output from WinHSPF and corresponding gage data.

1.   From the **Start** menu under **Programs**, select **BASINS** and then **WinHSPF**.

2.   From the **File** menu, select **Open** or click the "Open Existing Project" button, [icon].

3.   Navigate to *c:\basins\modelout\Westpat* and select "*WestPat.uci.*" Click **OPEN**.

4.   Click [icon] (View Output button).

       Note: The "GenScn Initialization from BASINS" window will appear. This form has six different tabs because many types of files can be viewed within the GenScn interface.
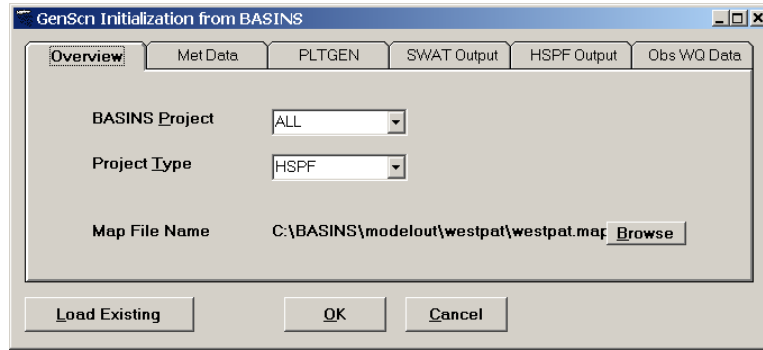
> **Note:** The "Overview" tab shows information about the GenScn project that is being created from the BASINS interface.

5.   When used from BASINS, by default the appropriate settings will be selected. Click OK.

The GenScn window will open. This window has six sections: Locations, Scenarios, Constituents, Time Series, Dates, and Analysis.
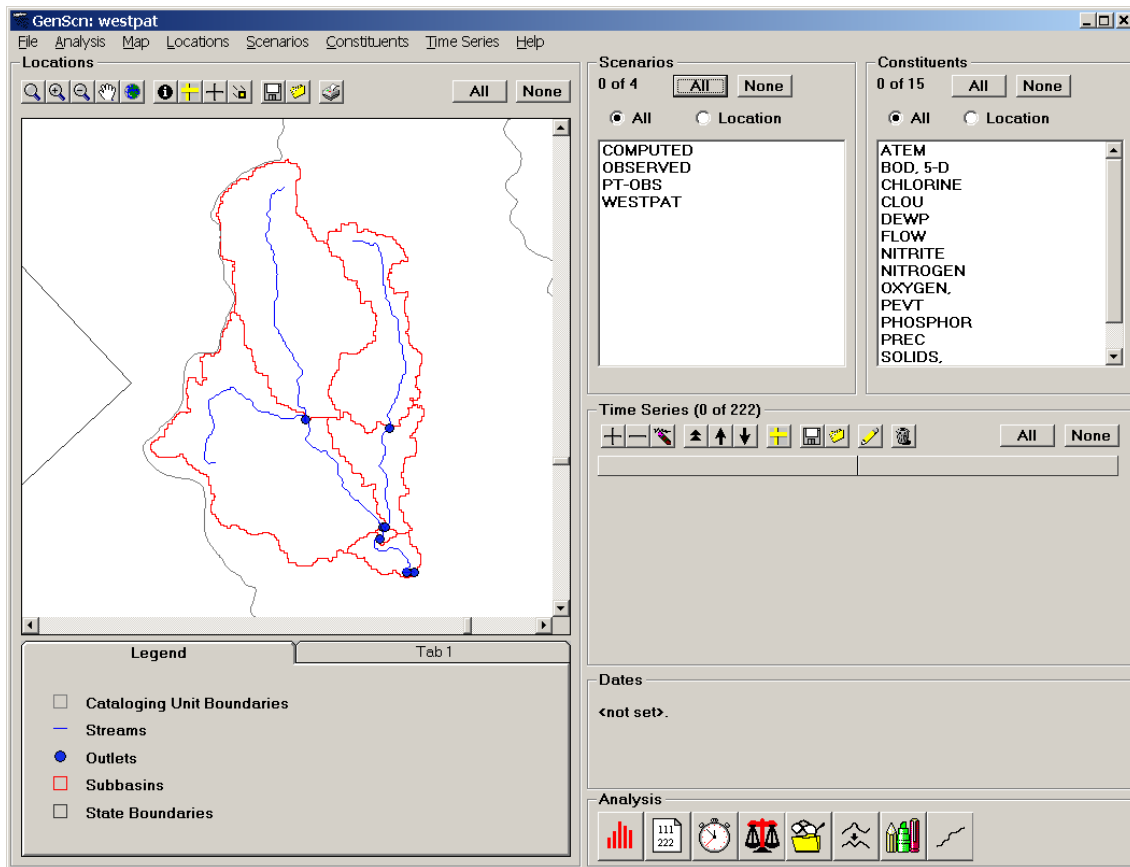
Recall that we want to look at a graph that includes both the model's flow simulation values and observed flow values for the same reach in order to ascertain how well we are simulating flow in Reach 5. We will graph both times series in the following steps.

3.   In the "Scenarios" frame, select "WESTPAT" and "OBSERVED."

4.   In the "Constituents" frame, select "FLOW."
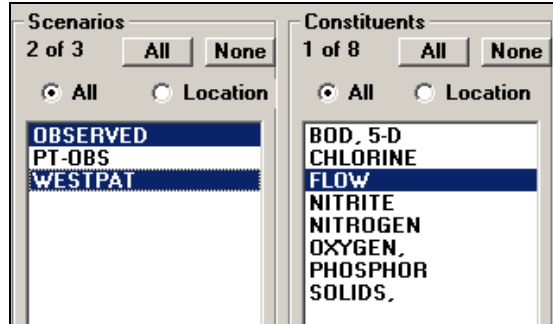
5.   In the "Time Series" frame, click ⊞ (Add to Time Series List). All time series for flow within the "WESTPAT" and "OBSERVED" scenarios will be listed in the Time Series section.

6.   In the "Time Series" frame, highlight the record that corresponds to the scenario name "WESTPAT." Press SHIFT and highlight the "OBSERVED" time series.
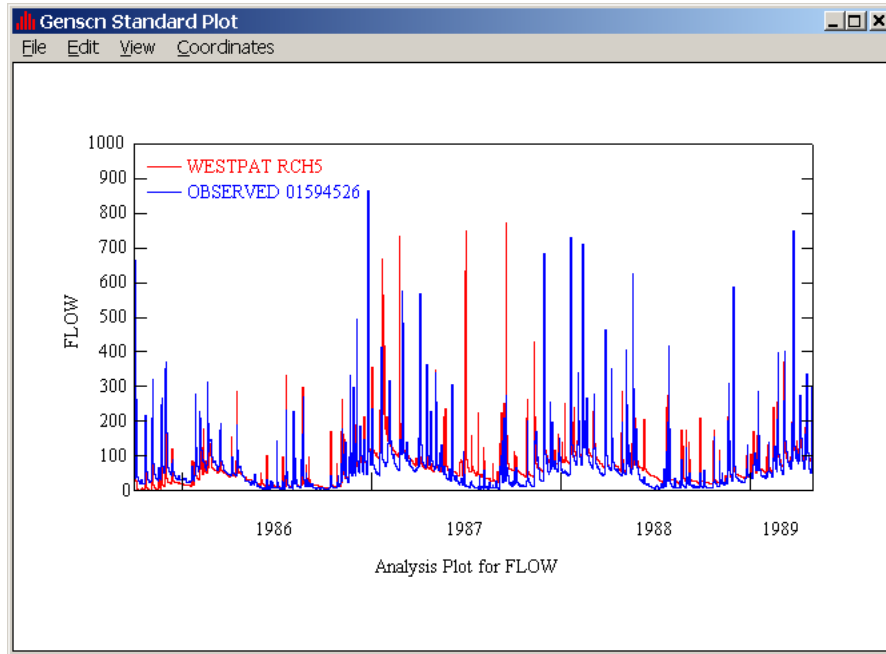

7.   In the "Analysis" frame, click 📊 (Generate Graphs).

8.   Make sure the "Standard" box is checked.
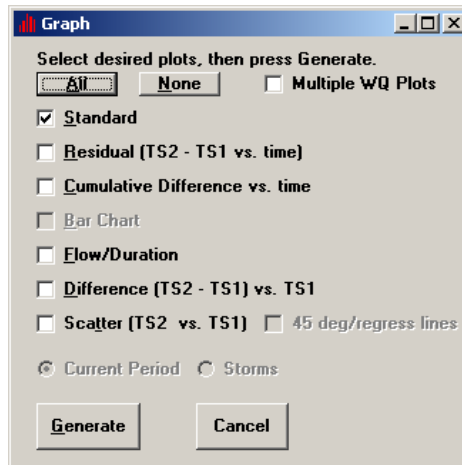
9.   Click GENERATE.   The plot window will open:

10.   After looking at the graph, exit the GenScn Standard Plot. Click CANCEL to exit the "Graph" window.

11.   Zoom in on smaller time periods by changing the start and end dates in the "Dates" frame. Repeat steps (6-9) to look at the graph of this time period.

12.   Exit any open graphs and close GenScn. You don't need to save any changes.

Special Note about HSPF Errors in WinHSPF:

From time to time changes made to the User Control Input (*.uci*) file through WinHSPF (or a text editor) are interpreted as errors the next time the *.uci* is opened in WinHSPF. This condition is indicated by a message similar to the following when opening a *.uci* file in WinHSPF:
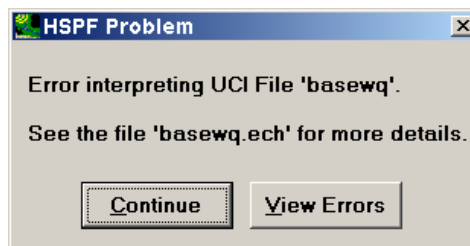
Details of the error encountered are written to the file known as the "Echo" file which is named with the same base name as the *.uci* file only with a *.ech* extension (e.g., *w_branch.uci* will have an "Echo" file named *w_branch.ech*). Clicking the "View Errors"

button will provide a view of the *.ech* file at the point of the first error. HSPF error messages are enclosed in asterisks as shown below:

The *.ech* file may also be opened in any text editor (such as Wordpad). Searching for the text 'Error' will take the focus to the portion of the file containing error messages. Once the error messages are found, the cause of the problems can be identified and resolved.

It should also be noted that it is also possible to encounter errors by HSPF during simulation that prevent the model from running to completion. If the simulation does not appear to run to completion (for example the output time series plots do not contain data for the full span of the run), the *.ech* file should be examined for error messages. The *.ech* files are generally located in the same directory as your *.uci* file.

Reference:

Duda, P., J. Kittle, Jr., M. Gray, P. Hummel, R. Dusenbury. *WinHSPF Version 2.0: An Interactive Windows Interface to HSPF (WinHSPF), User's Manual.* Washington, D.C.: U.S. Environmental Protection Agency, 2001.