DRAFT REPORT

WASH MODEL APPLICATION GUIDE

Prepared for
SFWMD

May 9, 2008

URS

URS Corporation
1625 Summit Lake Drive, Ste. 200
Tallahassee, FL 32317
12805298

TABLE OF CONTENTS

TABLE OF CONTENTS

1.1 PURPOSE OF DOCUMENT - RELATION TO THE OTHER WASH DOCUMENTS

The WaSh model is a coupled watershed water quality model (WaSh) originally developed with funding from the South Florida Water Management District (SFWMD) for simulating conditions in the St. Lucie Estuary (SLE) basins. WaSh is a time-dependent, coupled hydrologic and hydraulic simulation model. It includes representation of basic surface hydrology, groundwater flow, surface water flow, point sources and losses, and water quality fate and transport. It also includes many special features specifically required to simulate conditions in the SLE basins, such as irrigation demand and supply, high water table conditions, fully coupled groundwater and surface interactions, reservoirs and Stormwater Treatment Areas (STAs), and flow structures.

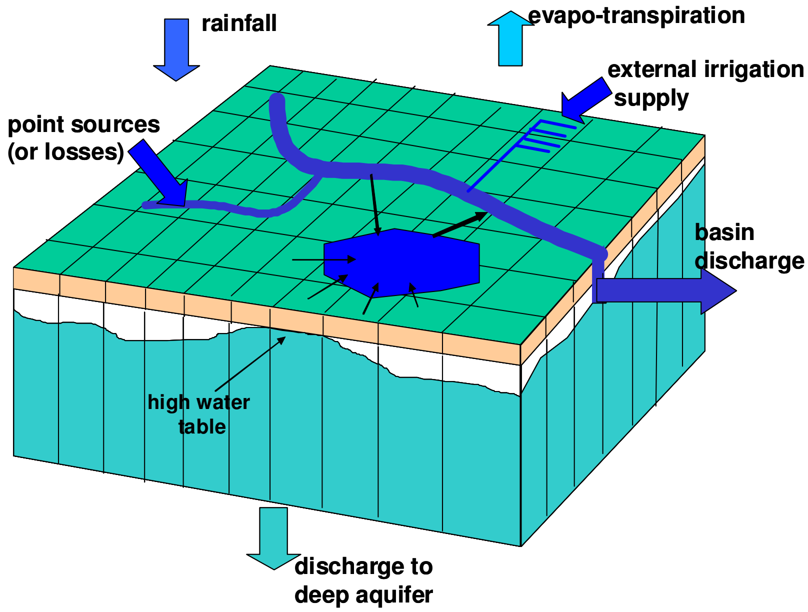
This **WaSh Model Application Guide** is one of three manuals comprising the full WaSh Model documentation. The three manuals are:

- The **WaSh Model User's Manual** which provides a detailed reference for each function and feature of the WaSh Model. It presents an introduction and overview of the model system, model capabilities and limitations, installation instructions, how to get started, detailed descriptions of each model component, data preparation, detailed model set-up, model parameters estimation, running the program, how to view model output, and some discussion about analysis of model simulation runs.
- The **WaSh Model Application Guide** contains a tutorial that demonstrates various aspects of the WaSh model application. Each example consists of a problem statement, data requirements, detailed model set-up steps, displays of key input and output, and discussions of important modeling aspects. By going through these examples, a less experienced user will be able to conduct basic WaSh model application.
- The **WaSh Model Theory Documentation** describes the theory and data requirements for the hydrological simulation (including water quantity and water quality) performed by the WaSh. The mathematic model equations used in WaSh are presented along with the assumption used in their derivation. Methodology, algorithm, and numerical scheme, matrix solutions used in the WaSh are explained. The document provides reasonable range of model parameters when applicable, and guidelines on model parameter estimation.

The **WaSh Model Application Guide** provides a tutorial that demonstrates a typical WaSh model application, from installation through model configuration, execution and post processing. The Application Guide is intended to provide hands-on training in which the user mimics the steps detailed in the Guide. The Guide starts with a basic hydrology simulation and then adds additional hydraulic features. With each successive modification to the model, the simulation outputs are processed so that the user can gain insight on the effects of the modifications. Basic and Advanced water quality simulation are then described.

1.2 BRIEF DESCRIPTION OF THE WASH MODEL

The WaSh model consists of four basic components: (1) a cell-based representation of surface hydrology, (2) a cell-based representation of subsurface hydrology (groundwater), (3) a reach network that represents streams, rivers, estuaries and canals, and (4) reservoirs and STAs. Each of these components is shown in Figure 1.1. In addition to providing a representation of surface hydrology, each cell can optionally store surface water in a system of tertiary canals. These canals receive direct surface runoff; convey water to the reach network, reservoirs and STAs; and interact with the groundwater (more information is provided in the runoff routing section).



1.1

The basin can be considered a bounded system with distinct sources and sinks. The sources and sinks, also identified in Figure 1.1, are:

1) Rainfall
2) Transpiration/Evaporation
3) Point sources or sinks
4) Percolation of a deep aquifer
5) External sources of irrigation water
6) Discharge from the basin

The first three items are user-specified input to the model, whereas the last three items are calculated internally by the WaSh model. External irrigation supplies are irrigation water that is supplied from outside the system, either from another basin or from the deep aquifer. The WaSh model calculates the external supply as the part of the irrigation demand that cannot be met with internally stored water.

A functional schematic of the WaSh model components and how they interact is shown in Figure 1.2. This figure also serves as a summary level block diagram of the WaSh model computer program. The computations are divided into four groups:

1) Hydrology calculations
2) Cell discharge routing

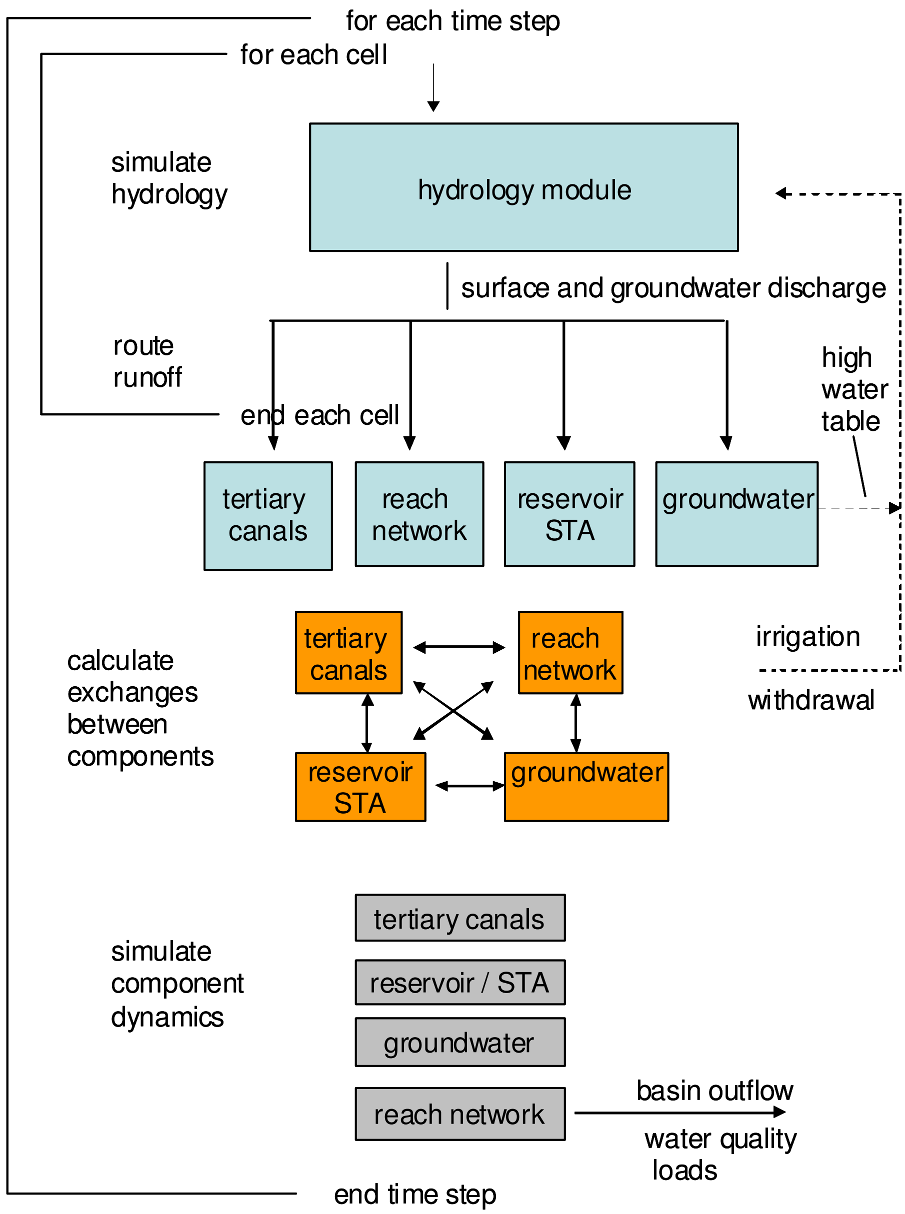
3) Component interactions
4) Individual component dynamics

For each time increment, the surface hydrology routine produces a surface and subsurface (percolation) discharge for each cell. The cell's percolated water is routed to the groundwater, and the surface discharge is routed to one of the receiving water bodies, i.e., either to the tertiary canals, reach network, or reservoirs and STAs. The decision as to where each cell's surface discharge is routed depends on the basin geometry and on user-controlled setup of the WaSh model. The surface hydrology simulation is based on the HSPF algorithms (Bicknell et al., 1997; Johanson et al., 1981).

There is some coupling between the cell hydrology routines and the other components, due to possible high groundwater conditions or irrigation demand. The high groundwater module will reduce percolation when the groundwater rises into the surface moisture. When a cell's irrigation module is active, it will sense deficiencies in the moisture zones and calculate an irrigation demand to replenish the deficiency. The demand is allocated by the user to tertiary canals, the reach network and reservoirs, as well as to external sources. The demand is extracted from each component during the component simulation.

After the discharges are routed, the exchange of water between components is calculated. All four components can interact with each other, via groundwater-surface water exchanges, pumping, and gravity flow.

Once the fluxes between components are calculated, the response of each component is determined. The groundwater model consists of an unconfined aquifer, intended to represent the surficial water table. The reach network model is based on the one-dimensional shallow water wave equation, and includes branching and representation of structural control on flow and stages. The tertiary canals are represented in each cell as a storage potential, with flow controlled by user-specified parameters describing weir configurations and pumping operations. The reservoirs and STAs are also represented using a storage potential with six different flow control options for controlling flow into and out of the facilities.



1.2

The basic time period for generating model output is one day, with flows, loads and other fluxes provided as daily averages. However, certain processes are simulated using a smaller time step in order to properly resolve the system dynamics. The cell hydrology calculations are simulated with a one-hour time step. Surface water dynamics in the reach network, tertiary canals and reservoirs and STAs are completed on a half-hour time step, and the cell-to-cell groundwater flows are calculated with a daily time step. However, the groundwater exchanges with surface water (reservoirs, tertiary canals and the reach network) are calculated using a half-hour time step. Flux rates between components using different time steps are aggregated, as needed, in order to preserve mass balance.

The water quality component of WaSh is based on the time-dependent scalar transport equations and is directly coupled to the flow model. There are two options for simulating water quality, the basic option (BWQ) and the advanced option (AWQ). The basic option consists of simulating the runoff and transport of five water quality constituents, Total Suspended Solids (TSS) and dissolved and particulate nitrogen and phosphorus. The constituent transport is based on mass conservation, but includes first-order decay and settling and re-entrainment of TSS and particulate fractions of nitrogen and phosphorus.

The AWQ option is based on the non-linear dissolved oxygen (DO) balance and eutrophication model as described in the WASP model documentation (Ambrose et al., 1988). There are eight constituents (state-variables): ammonia, nitrate, inorganic phosphorus, phytoplankton carbon, carbonaceous biochemical oxygen demand, dissolved oxygen, organic nitrogen, and organic phosphorus. The run-off and transport portion of the advanced water quality option is identical to that of the basic option. The primary difference occurs in the surface water bodies, for which the transformations between state-variables are also simulated in the AWQ option. Also, each reach in the reach network can be divided into an upper and lower section. Vertical diffusion between the upper and lower sections can be simulated also, by specifying a diffusion rate. This capability was added to improve representation of conditions in which there are significant vertical variations in the water quality constituents.

The WaSh model also supports the implementation of numerous Best Management Practices (BMPs) used to control the flow and water-quality characteristics of the basins. The BMPs are divided into two general categories, explicitly represented, and implicitly represented BMPs. Explicitly represented BMPs include structures that are included directly in the drainage system. These consist of detention and retention basins and certain irrigation practices. Implicitly represented BMPs include a wide variety of practices, ranging from filter strips to grazing practices, and are represented as modifications to model parameters controlling runoff quantity and quality.

The model is supported by a Graphical User Interface (GUI) that is developed as an ArcView extension. The GUI provides functions for configuring the model using “raw” data. Once a model is configured, the GUI provides the user with functions to modify the model, execute simulations and post-process model outputs.

1.3 DESCRIPTION OF THE PHYSICAL SYSTEM USED IN THE APPLICATION GUIDE

The Application Guide for WaSh GUI will use a section of the St. Lucie Estuary (SLE) Watershed for demonstration purposes. All configuration and forcing data has been procured from the South Florida Water Management District’s GIS Mapping database or DBHydro Environmental database and provided as part of the tutorial. In the tutorial, the watershed is initially represented according to the existing system. However, hypothetical situations are later created in the model to illustrate certain WaSh features. Therefore, it should be noted that the existing system of the actual SLE watershed may differ slightly from the modeled watershed in this guide.

1.3.1 Regional Setting

The St. Lucie Estuary (SLE) is located near the Martin and St. Lucie County line on the east coast of south central Florida about 30 miles from the eastern edge of Lake Okeechobee. The St. Lucie Estuary is the largest brackish water body on the east coast of Florida. The main branch of the estuary spans from the St. Lucie Inlet at the junction with the Indian River Lagoon and spreads northwest for 3.5 miles. Just past the A1A Bridge the estuary makes a turn due west, until it hits the Roosevelt Bridge at US1 2.5 miles beyond. At US1, the estuary branches out into the North and South Fork. The estuary width spans from 1000 meters at the inlet to nearly 2000 meters at east of the Roosevelt Bridge. The North Fork averages approximately 1200 meters in width for a length of 4.5 miles, while the South Fork averages 700 meters width or three miles. Channel depths of the estuary range up to 5.4 meters, but the average depth of the main channel is approximately 3 meters.

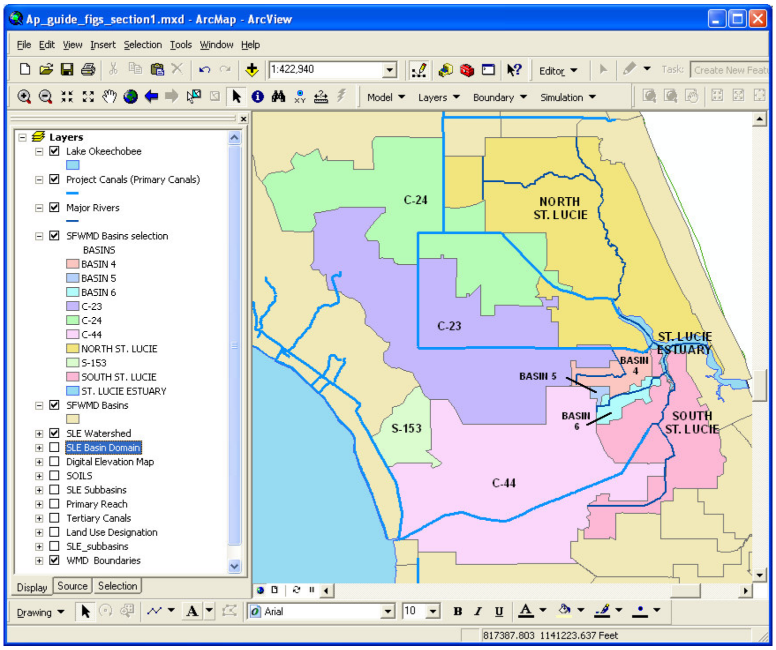
The inner SLE is comprised of the North Fork and the South Fork of the St. Lucie River, and has a total surface area of approximately 6.4 square miles. The two forks converge to form a single middle estuary with a surface area of 4.7 square miles. Five tributaries to the estuary provide drainage for a watershed that encompasses 827 square miles (SFWMD data).

Although the SLE was once a fresh water system, the opening of the St. Lucie Inlet in 1892 instigated the switch to a brackish system. In 1937 following the completion of the Okeechobee Waterway by the USACE, the SLE was connected to Lake Okeechobee via a 152-mile-long, cross-state canal system. The connection of the lake to the estuary allowed the USACE to drain water from the lake during times of excessive rain to prevent local area flooding. Consequently, during periods of high discharge from the lake, the estuary water quality is degraded by a lowering of salinity, an increase in turbidity, and an increase in nutrients from the lake. Conversely, flow in the Okeechobee Waterway can be reversed, directing flow from the canal into the lake, during times of draught. During these times estuary levels may drop, further stressing the system.

Further degradation of the estuary has been precipitated by both the continued urbanization of the coastal regions of St. Lucie and Martin Counties and by the increased agricultural activities in the western regions of the watershed. Agricultural activities contributing to the declining health of the system include water flow changes created by the creation of secondary and tertiary canal systems for use in irrigation, the introduction of nutrients via fertilization, and the increased volumes of sediment in the agricultural regions.

1.3.2 Basin Physical Characteristics and Model Configuration

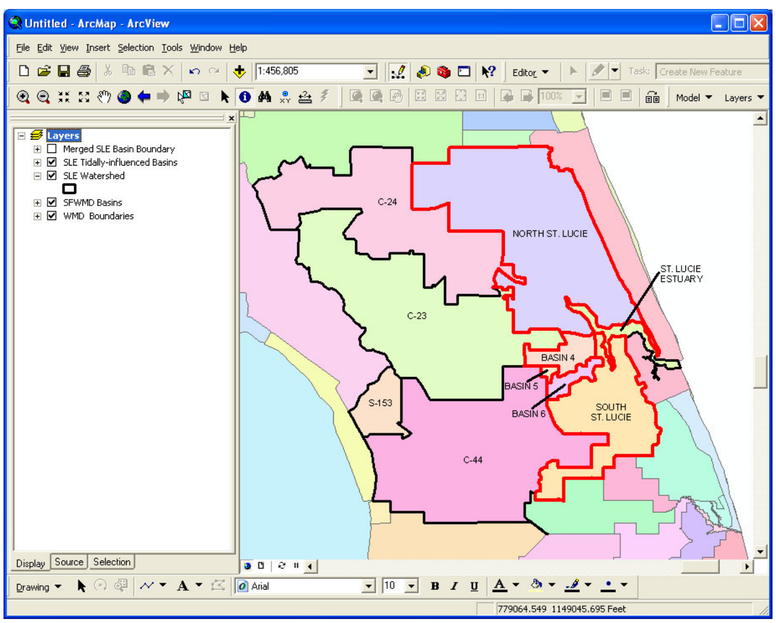
Watershed modeling of the SLE system is dependent on the accurate representation of the many physical components present within each drainage basin. A map of the SLE and its corresponding watershed (SLEW) is shown in Figure 1.3. The SLEW is composed of nine basins that drain into the SLE: C24, C23, C44, S153, Basin 4, Basin 5, Basin 6, North Fork, and Tidal St. Lucie.



1.3

The three major upland basins contributing to the SLEW are the C24, C23, and C44 basins. Each of these basins is dominated by a project canal which collects both tertiary canal system discharges and over-land sheet flow runoff discharges from the basin and directs flow through a control structure at the basin outfall. The basin outfalls drain to the SLE. The project canals and control structures are identified in Figure 1.3. The remaining basins contain rivers and tributaries that are tidally-connected to the SLE.

During this tutorial, the WaSh model will be configured to represent only the tidally-connected basins. The outline of the model area is shown in red in Figure 1.4.



1.4

The SFWMD has been collecting daily flow data at the C24, C23 and C44 basin outfall structures for over 40 years. Monthly water quality data has also been collected for over 25 years. These data are used in the modeling to represent the upland basin discharges to the SLE. They will be added as point sources and are discussed more in Section7.0

Tidal boundary conditions will be applied at the downstream end of the model, which is located at the eastern end of the middle estuary. Two tide gage stations were located in that vicinity with data available for the period of 2002-2005. One station was located on the A1A Bridge near the St. Lucie Inlet and one was located on the Hwy. US1 Bridge near the branch of the North and South Fork (Figure 1.3). Data was collected by the USGS at 30-minute intervals (SFWMD personal communication). These data are used for setting the downstream tidal boundary condition in the example application.

Other key basin features that will be represented in the example WaSh application are tertiary canals and irrigation practices. All of the SLE basins have either all of the basin or significant portions of basin drained by a canal system. The canal system can be divided into three components: a tertiary canal system, a secondary canal system, and a primary (project) canal system. The tertiary system consists of dense networks of narrow, shallow canals within the agricultural lands used to control soil water levels and provide water for local irrigation. Water levels in the tertiary system are controlled by a series of weirs operated by the local land owners. Little stage data is available regarding water levels, volumes or flows of secondary or tertiary canals. The tertiary canals are linked to a secondary system of canals, which were built to provide a connection between tertiary systems and the project canals. The influence of the tertiary canals in the SLE drainage basins is explicitly represented in the example application.

Other unique characteristics of the system are the irrigation practices. Water for irrigation is drawn both from the tertiary canal systems and from the Floridan aquifer. It is estimated that about 28% of the irrigation water was derived from subsurface sources. However, the rates, amounts, and timing of the irrigation are not well known, nor are the relative volumes of water drawn from either the canals or from the Floridan. This is an important aspect, since water drawn from the Floridan is essentially an additional input of water into the system that will affect the average flow rates through the canal system. The irrigation practices are represented in the WaSh example application using the irrigation module.

Finally, there is only one major structure within the tidally-influenced portion of the SLEW, the Gordy Road structure. The structure components and operational guidelines area available from the SFWMD for this structure and are used to configure the structure in the WaSh example application.

The WaSh Model operates as a Graphical User's Interface toolbar within ESRI's ArcGIS software. Prior to installing the WaSh GUI, ArcGIS 9.2 must be installed on the PC to be used for modeling. The Spatial Analyst extension must also be installed in order for ArcMap to disseminate the vertical elements of drainage. More information about the GIS software package can be found on ESRI's website at:

http://www.esri.com/software/arcgis/index.html.

Once ArcGIS has been installed, the WaSh GUI software is ready to be installed.

2.1 MODEL INSTALLATION

2.1.1 Operating System Requirements

The WaSh Model is accessed and operated through a Graphical User's Interface (GUI) that functions within ArcMap 9.2 as a toolbar. ArcMap 9.2 must be installed prior to installation and operation of the WaSh Model.

Basic operating system requirements for ArcMap 9.2 are:

- CPU: 1.6 Ghz Pentium 4 or better
- Memory: 1 GB Minimum
- Disk Space: 1.2 GB Minimum
- Spatial Analyst is also required for certain features to function properly, such as the topographic processing using elevation raster files

The Wash GUI itself has the additional requirements of two MBs of disk space on the local hard drive. The Wash GUI extension will install and operate within ArcMap 9.1, but it has not been fully tested and is not supported.

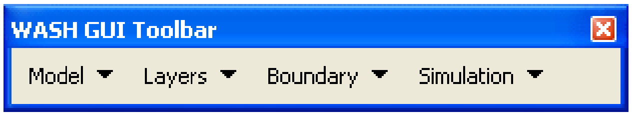
2.1.2 Self-Extracting Executable

The WaSh Model GUI and Model Code are typically delivered in a zip file that contains two self-extracting files: Setup.exe and WASHGUI_SETUP.msi. To begin installation, run Setup.exe. This operation will verify that .NET 2.0 is installed on the PC. If .NET 2.0 has not been installed, the installation will be initiated for .NET 2.0 and the user will be instructed on the installation procedure. If the .NET 2.0 is currently installed, the installation of the Wash GUI will begin. The only user input required during the installation is the specification of the file for the model root directory. The default path in the Program Files directory is recommended, but any path can be specified.

After installation is complete, a folder named "SeedFolder" will reside in the WaSh GUI installation folder. The SeedFolder contains files vital to the operation of the Wash GUI. Files with the .lyr extension control how different features are displayed on the map. The WashGUI.config file contains several settings. The SeedTables.mdb file contains default values for many of the Wash Model parameters. It is recommended that you do not change these file names.

2.1.3 Initializing WaSh Toolbar

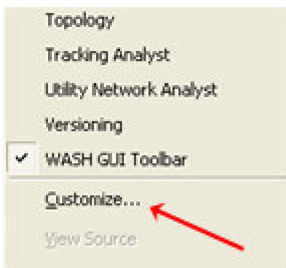
After installing the Wash GUI and opening an ArcMap session, the WaSh GUI toolbar should be visible and appear as:



2.1

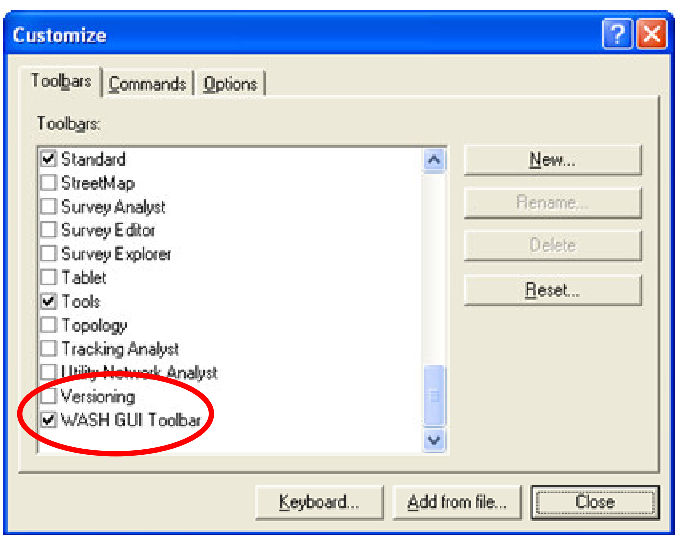
The toolbar can be removed from the center of the layout by adding it to the toolbars along the frame of the ArcMap screen.

If the Wash GUI toolbar is not visible, the list of available toolbars can be accessed by right clicking anywhere in the toolbar area. Activate the GUI toolbar by checking the associated box next to the "WASH GUI Toolbar" on the toolbar list.



2.2

If the WaSh GUI Toolbar is not on the list, then click "Customize..." and in the dialog that appears, click the "Add from file..." button. Browse to the folder where Wash GUI is installed and select "WashGUI.tlb". Once the correct path has been identified, make sure "WASH GUI Toolbar" is checked in the list. The Wash GUI Toolbar should now be visible in the ArcMap session.



2.3

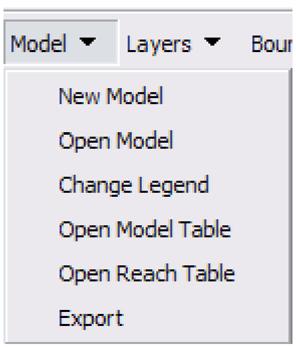
2.2 GUI VIEWS AND APPEARANCE

This section gives a general introduction to the WaSh GUI menus, legends and color-coding procedures.

2.2.1 Navigating the GUI Menus

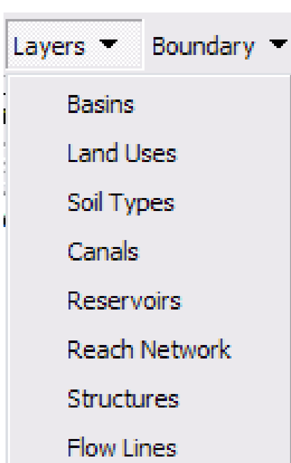
The main GUI toolbar comprises four menu options as shown in Figure 2.1. The Model menus allow the user to open and save models, display features and edit model parameters. The Layer menu options allow the user to configure a new or existing model. The Boundary menu options allow the user to specify model forcing conditions, such as rainfall and evaporation, tidal elevations and flows. Finally the Simulation menu provides options for controlling the model simulation. The contents of the four toolbar menus are summarized below.

From the Model menu, shown below, a New Model can be created and a new grid can be generated. An Existing Model can also be opened from this menu. The model grid appearance can be altered using the Change Legend feature in this menu. Additionally, the Model and Reach tables, which contain all of the HSPF model parameters affecting the simulation of hydrology, hydrodynamics, and water quality, can be accessed from this menu. Lastly, the Export feature can be used to export table parameters. Some of the menu options may not be available until a model is fully configured and operational. Certain functions may be disabled or "grayed out" until prerequisite information is available. For example, you will not be able to access the change Legend option until a New Model is created or an existing model is opened.



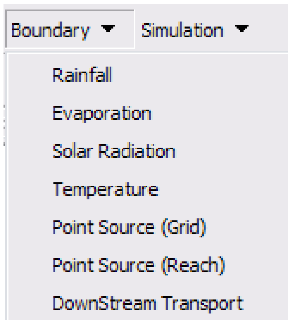
2.4

The Layers Menu offers the following features required for the configuration of a model. From this menu, a new model grid will be attributed with the physical parameters of basin and subbasin identification, land use, soil types, and surface hydrology (tertiary canals and primary reach). Optional configuration data, such as the addition of structures and reservoirs will be added from the layers menu. Finally, the flow pathing for surface and ground water hydrology will be established from this menu.



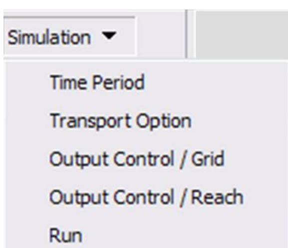
2.5

The Boundary menu allows for the input of forcing data or external data which drives the mechanics of the system, such as Rainfall, Evaporation, Solar Radiation, and Surface water temperature. Both Grid and Reach Point Source data will be added to the model from the Boundary menu. The last option on the menu is the Downstream Transport option, which is used to establish boundary conditions for either structurally-controlled or tidally-influenced systems.



2.6

The fourth menu, the Simulation menu, allows the user to set the time period for the simulation. The user can also define the type of modeling to be simulated, such as hydrology or water quality and hydrology, using the Transport Option. Output control is also defined from this menu. Any grid cell or reach segment will have an output generated during the simulation. Any cell or reach segment identified in the Output control option will have the output data written to a comma-delimited text file which will be placed in the model folder. Finally, simulations can be launched from this menu.



2.7

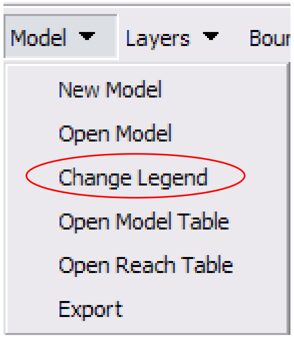
Further details of each menu option and associated sub-menus are described and discussed as they are used throughout the tutorial.

2.2.2 Changing Legends

During the configuration of a model, many screen views of the model grid and other features will be required. The GUI generally will display the most relevant view for the intended operation. However, there will be times when the user may wish to change the appearance of the screen to aid in the model configuration or execution. For instance, in the situation where the user wishes to select certain land use cells within a geographic area, the user may wish for the display to provide color coding based on cell land use and enhance the cell color coding to highlight level three land use categories. An experienced ArcGIS user can likely navigate through the layer data and make the desired changes. However, certain often used and helpful operations have been made available to the less experienced user via specialized GUI menus.

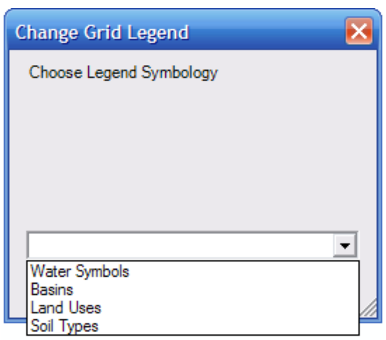
The basic model view consists of the grid cells, upon which other model objects such as reservoirs, structures and the primary reach network can be displayed. The grid cells can be color coded to show the spatial distribution of any one cell attribute at a time. The GUI menu option used to change the display is the Change Legend tool which is access from the Model menu. The GUI menu options to show other model features are available under each features menu in the Layers menu.

By selecting the Change Legend tool, as indicated in the figure below,



2.8

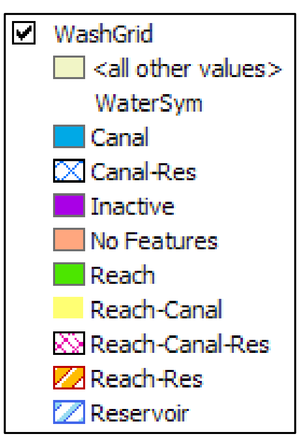
a legend coding option can be selected for display of the grid.



2.9

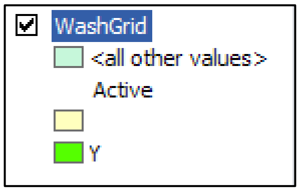
Four options are currently available for setting the model grid display: water features, basin, soils and land use. Below are the legend examples of each default color-coded legend option.

When the Water Symbols legend is selected, the display will color code the cells according to their "type" specification. The "type" attribute (Section ?.? of the User's Manual, Section ?.? of the Theory Manual) specifies how surface and groundwater are routed to and from the cell. Below is an example of the Water Symbols legend.



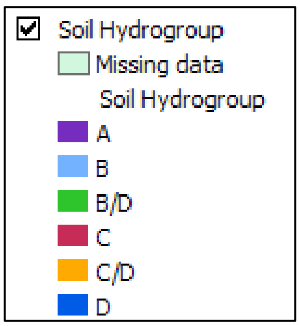
2.10

The Basin legend options color codes the cells according to sub-basin designation. An example of the legend is provided below.

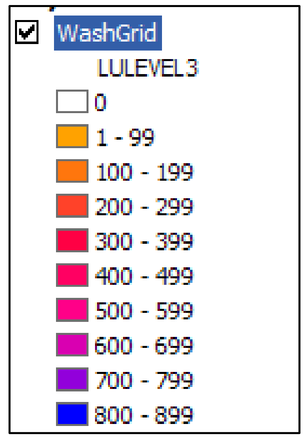


2.11

The Soils Type and the Land Use legend color code the grid cells according to the soil type attribute and the land use attribute. Examples of each legend are shown below.



2.12



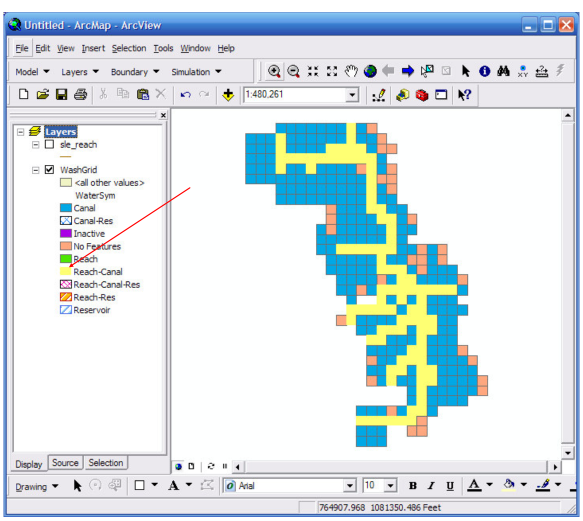
2.13

The legends for the other model features are described later in the tutorial as each feature is configured and edited.

2.2.3 Assigning New Symbols and Color Codes

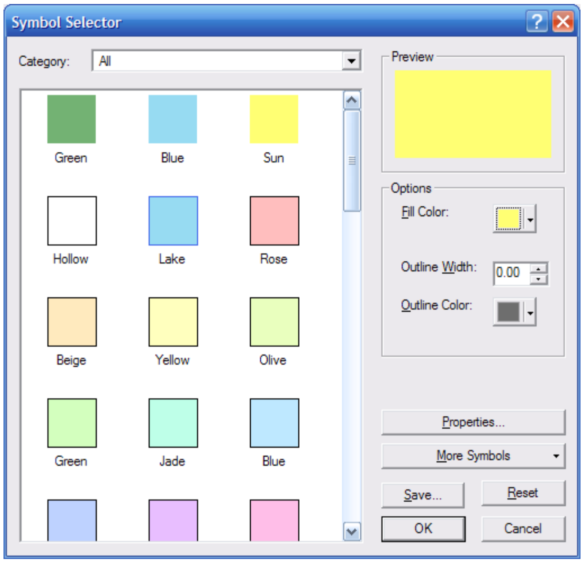
In addition to setting the cell attribute that is color coded and displayed, the user may also wish to change the default color codes. These procedures can be done with the basic ArcGIS tools and are not directly supported with the WaSh GUI documentation. However the general ArcGIS steps involved in making the changes are briefly reviewed here so that inexperienced ArcView users can use these tools during the tutorial.

If an individual color is to be changed, double-click on the symbol in the Layers list.



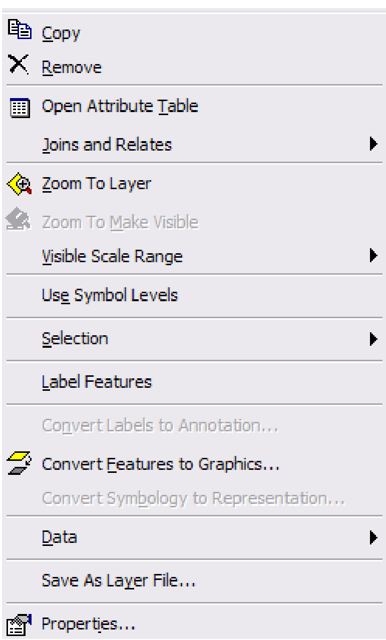
2.14

The following Symbol Selector option screen will appear. Changes can be made through this screen through various drop-down menus and buttons. For grid polygons, the main symbols that change are Fill Color (red arrow) and outline color (blue arrow).



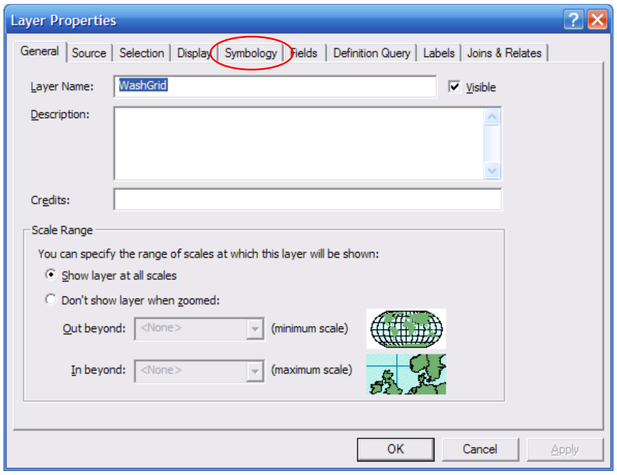
2.15

If more than one legend item needs to be changed in a single layer, the symbology for an entire layer can be accessed at once by right clicking on the grid file name, for example. The Properties option can be selected from the menu.



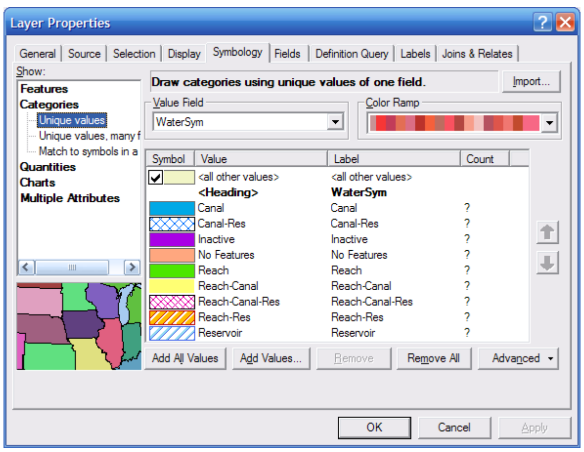
2.16

Select the Symbology tab from the tab options below.



2.17

The following screen lists the default Water features legend. To change any color, double click on the symbol and choose a different color.



2.18

Editing polyline or point files works in the same manner. One useful legend change is to change the Flow Lines symbol from simple lines to arrows. Flow paths are easier to define when the direction is indicated by the arrow. A more detailed description of the change is discussed in Section 3.5.

For more detailed instruction on Symbology editing, reference ESRI's ArcGIS documentation.

2.3 REVIEW OF INPUT FILES

The WaSh model requires various input data sets in order to configure and operate the model. The data are divided into two general categories: configuration data and forcing data. Configuration data consists of physical parameters that describe the system, such as topography and land use. Typically, the configuration data represents spatial variations and is considered time-invariant. Forcing data consists of input data which acts upon the physical system to alter flow and water quality. Typically, forcing data will be time varying and constant in space. However the spatially variable rainfall data may also be used with the WaSh Model.

Each of the files used in the tutorial are described below. They are all provided in digital form as part of the tutorial package. The user is encouraged to view the content of these files either in ArcGIS or in ASCII text editors.

2.3.1 GIS Shapefiles for Model Configuration

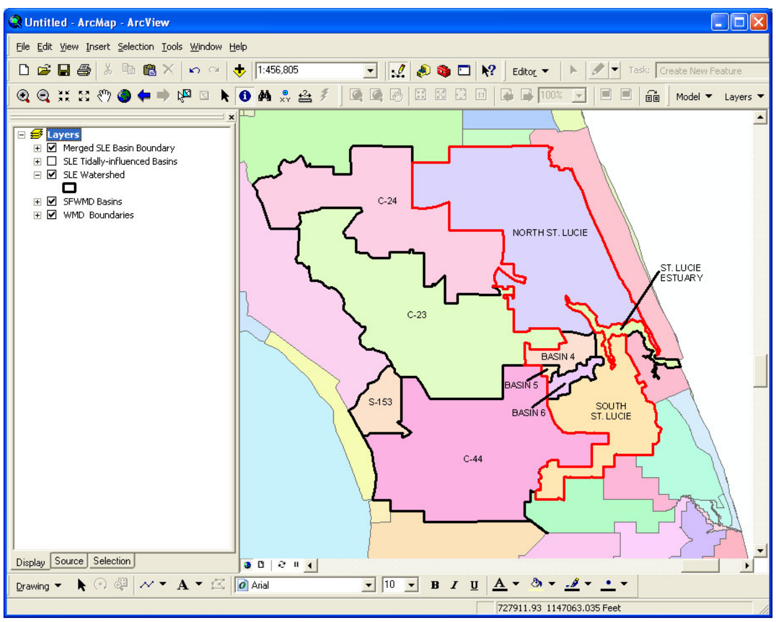
The physical characteristics of the basin are represented in the WaSh model on a uniform Cartesian grid. The spatial distribution of primary basins, secondary basins, land-use, soil type, topography, canals, rivers and streams, reservoirs and control structures are required to configure each basin model. The spatial data must all be in the same projection (i.e. state plane, local or UTM), with units of feet.

2.3.1.1 Model Domain: Watershed Boundary

The configuration file (i.e. shapefile) containing the watershed boundary is: SLE_basin.shp

This file contains one polygon that was created by merging B4, B5, B6, North Fork and South Fork basins into one polygon. As an alternative approach to providing the information, the user could supply each basin in a separate shape file and load then all during the configuration process.

When viewed in ArcGIS, the basin boundary will appear as the red outline in the Figure below.



2.19

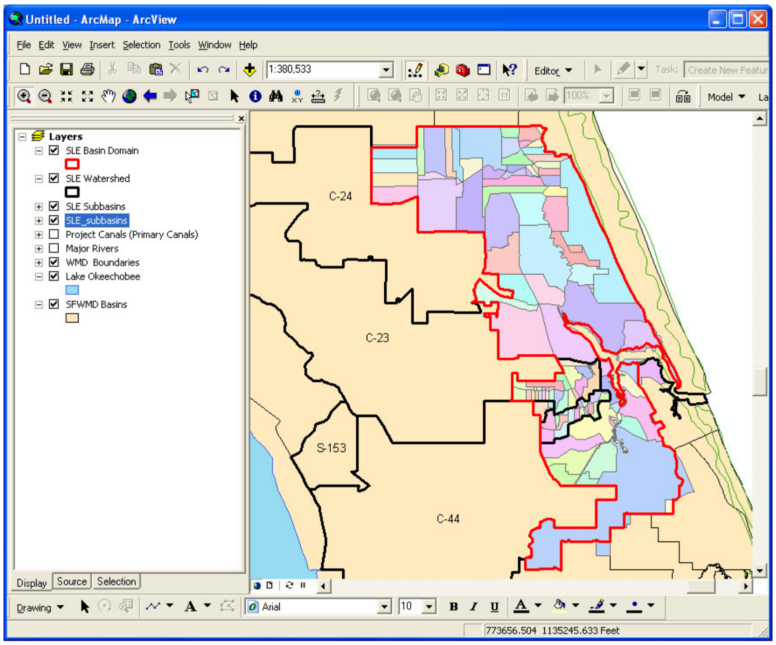
2.3.1.2 Secondary Basins

The configuration file (i.e. shapefile) containing the watershed boundary is: SLE_subbasin.shp

This file contains polygons for each of the sub-basins in the B4, B5, B6, North Fork and South Fork basins and their associated subbasin boundaries, as defined by SFWMD. Unlike the basin file(s), only one sub-basin file can be provided to the GUI, and sub-basins from all basins represented in the model must be merged into one shapefile.

If the user is not interested in attributing the model grid cells with the sub-basin name, then the basin shapefile can be used in place of the subbasin shapefile and each cell's subbasin name will be the basin name.

When viewed in ArcGIS, the subbasins will appear as the colored polygons within the red outline in the figure below.



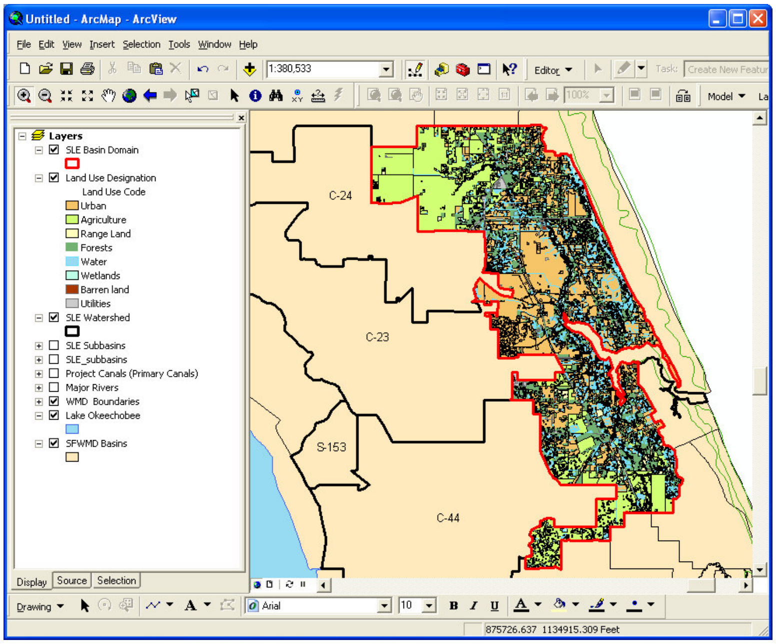
2.20

2.3.1.3 Land-use

The configuration file (i.e. shapefile) containing the watershed boundary is: SLE_landuse.shp

The attribute table associated with the land use polygon in the GIS shapefile must have the 3-digit numeric land use code assigned for each land use polygon. The WaSh model represents land use at a FLUCCS Code Levels 3. However, as land use is commonly designated with the 4-digit code (i.e. Level 4), the use must add a field or column to the shapefile's attribute table before use, rounding the FLUCCS code to a 3-digit code during processing. The user is expected to know the field name containing the FLUCCS code. In this tutorial the field name is: FLUCCSCODE_LEV3.

When viewed in ArcGIS, the landuse data will appear as the colored polygons in the figure below.



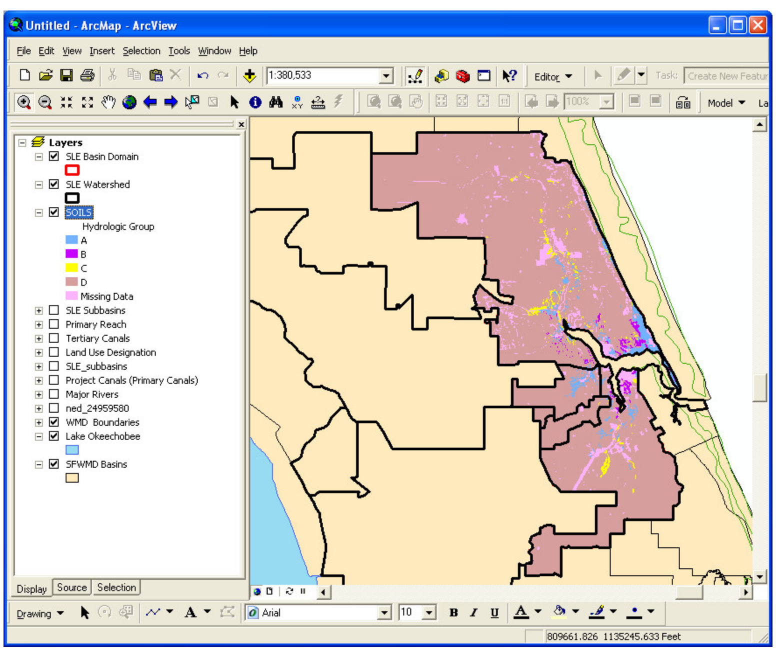
2.21

2.3.1.4 Soils

The configuration file (i.e. shapefile) containing the soils data is: SLE_soils.shp

Each polygon in the soils shapefile has one of the USGS hydro-group specification of A, B, C, D A/B, and C/D associated with it. The user is expected to know the field which contains the hydro-group designation so that it can be identified during the configuration process. In this file it is called "HYDGRP_C".

When viewed in ArcGIS, the soils data will appear as the colored polygons in the figure below.

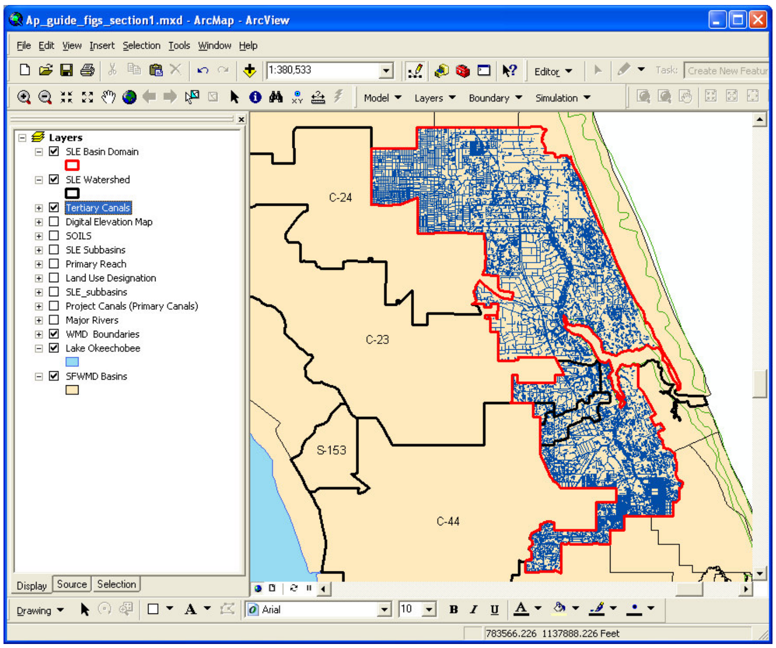


2.22

2.3.1.5 Surface Hydrology

The configuration file (i.e. shapefile) containing the tertiary canals is: SLE_tertiarycanals.shp.

This file contains polyline features representing the tertiary canal systems in the B4, B5, B6, North Fork and South Fork basins. During the model configuration, the length of each line within a cell will be tabulated and recorded as a cell attribute. The file was created by acquiring a hydrography file for the SFWMD GIS data base and then filtering out miscellaneous polylines included in the primary reach network (see Section 2.3.1.6of this tutorial). When viewed in ArcGIS, the tertiary canal data will appear as the colored polylines in the figure below.



2.23

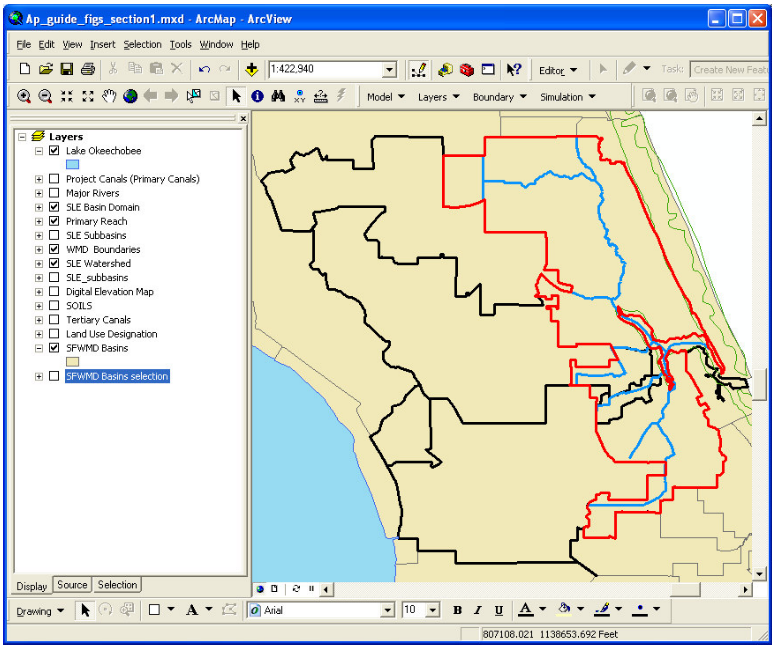
2.3.1.6 Primary Reach Network

The configuration file (i.e. shapefile) containing the primary river and canal system is: SLE_reach.shp.

This file contains polyline features representing the major rivers, canals and the centerline of the estuary in the B4, B5, B6, North Fork and South Fork basins. The file was created by acquiring a hydrography file for the SFWND GIS data base and then filtering out miscellaneous polylines that were included in the tertiary canal system.

The designation of the hydrography as part of the tertiary canal system or the primary reach network is the discretion of the user and the Model Theory Manual (Section ?.?) should be consulted for guidance on using these features.

When viewed in ArcGIS, the primary reach data will appear as the colored polygons in the figure below.



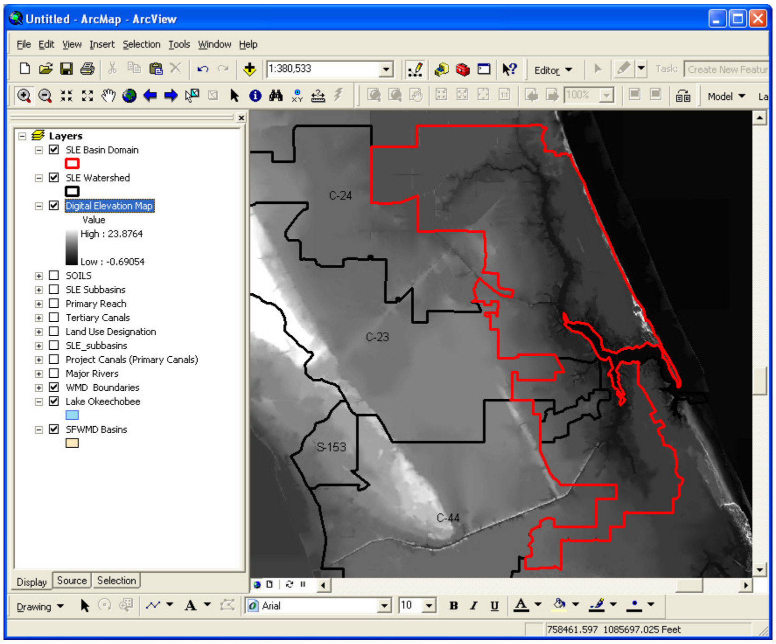
2.24

2.3.1.7 Topography

The configuration files containing the DEM topography data are a collection of files all with the base name: SLE_topo_sp. Inside the configuration folder, a subfolder named SEL_DEM contains all of the required files. While in ArcGIS, selecting the first SLE_topo_sp file listed will open the raster, which actually resided in another subfolder named SLE_topo_sp within the DEM folder.

This file contains a raster grid with elevation data in feet at the centroid of each grid cell. The vertical datum of this file essentially sets the project datum and all other vertical information, such as weir crest elevations and tidal elevations should be referenced to the same datum.

When viewed in ArcGIS, the topography data will appear as a grayscale image in the figure below.



2.25

2.3.1.8 Reservoirs

Reservoirs can be configured in the WaSh model in two ways, either by providing a polygon shapefile with the footprint of the reservoir, or by drawing it on the model grid using GUI tools. This last approach is intended to allow the user to quickly experiment with alternative reservoir designs without the need to geo-reference the polygons in shapefile. In the tutorial, both approaches are used. The configuration file (i.e. shapefile) containing the reservoir footprint is: SLE_reserviors.shp

In addition to the reservoir footprint, other attributes of the reservoirs such as their lining characteristics, height and operational features need to be specified. These data should be collected prior to configuring the WaSh model. In this tutorial, the data are provided later, in the section on adding reservoirs.

2.3.2 Time Series Files for Model Forcing

Model forcing data consists of input data which acts upon the physical system to alter flow and water quality outputs. Forcing data includes meteorological data such as rainfall, evapo-transpiration, radiation, temperature, point sources and boundary conditions. Some forcing data is required for a simulation, whereas some data is optional and depends on the water flow and water quality options used in the WaSh model configuration.

2.3.2.1 Rainfall for All Models

The time series file for providing rainfall data is: SLE_raindata.txt. It is an ASCII tab-delimited text file with columns for year, month day, hour and rainfall in inches/hour. Details of the required file format and period of record are provided in the User's manual in Section 3.2.1.1.

The data in this example file extends from 1/1/1993 to 1/1/2005. Thus, the period of the simulation must be within these limits. However, other time series input files may further limit the simulation period.

2.3.2.2 Evapotranspiration for All Models

The time series file for providing evapo-transpiration (ET) data is: SLE_evapdata.txt. ET is also reported as an ASCII tab-delimited text file with columns for year, month day, and potential evaporation in inches/day. Details of the required file format and period of record are provided in the User's manual in Section 3.2.1.2.

The data in this example file extends from 1/1/1965 to 1/1/2005. Thus, the period of the simulation must be within these limits. However, other time series input files may further limit the simulation period.

The daily potential ET values from this input file are modified in the WaSh model by a land-use dependent coefficient to represent variations in leaf cover and soil conditions for each land use. These values are accessed via the WaSh GUI edit menus are described in Section 6.1.5 of the User's Manual.

2.3.2.3 Solar Radiation for Water Quality Simulations

The time series file for providing daily solar radiation data is: SLE_total_radiation.txt. It is an ASCII tab-delimited text file with columns for year, month day, and solar radiation in Langely's/day. Details of the required file format and period of record are provided in the User's manual in Section 3.2.2.1.

The data in this example file extends from 1/1/1993 to 1/1/2005. Thus, the period of the simulation must be within these limits. However, other time series input files may further limit the simulation period.

The daily solar radiation is mapped to a diurnal variation using parameters that specify the seasonal sunrise and sunset time suing a half-sine function.

2.3.2.4 Temperature for Water Quality Simulations

The time series file for providing daily water temperature data is: SLE_temperature.txt. It is an ASCII tab-delimited text file with columns for year, month day, and temperature in degrees Celsius. Details of the required file format and period of record are provided in the User's manual in Section 3.2.2.2.

The data in this example file extends from 1/1/1993 to 1/1/2005. Thus, the period of the simulation must be within these limits. However, other time series input files may further limit the simulation period.

2.3.2.5 Reach Point Source Data

Three files are provided for specify source data to reaches. They are: C24_discharge.txt. C23_discharge.txt, and C44_discharge.txt.

These files are ASCII tab delimited text files with columns for year, month, day, flow and then eight columns for the concentration of each of the eight advanced water quality constituents. Details of the required file format and period of record are provided in the User's Manual in Section 3.2.3.2.

These files are intended to represent the upper basin discharges at the control structures. They contain both the flow and water quality loads based on measured data at those locations. The presence of the water quality concentrations is only required for the Advanced Water Quality Simulation (Section 7.0). If only hydrology and hydrodynamics were being simulated (Section 4.0), they would not be needed in the input files, and if present, the data in the advanced water quality columns would not be used in the simulation.

2.3.2.6 Downstream Tide Boundary Condition

The time series file for providing tidal elevation data at the downstream boundary is: tidalbc.txt. It is an ASCII tab-delimited text file with columns for year, month day, hour and tidal elevation in feet referenced to the project datum. Details of the required file format and period of record are provided in the User's manual in Section 3.2.3.3.

The data in this example file extends from 1/1/1993 to 12/31/2005. Thus, the period of the simulation must be within these limits. However, other time series input files may further limit the simulation period.

2.3.2.7 Downstream Water Quality Boundary Condition

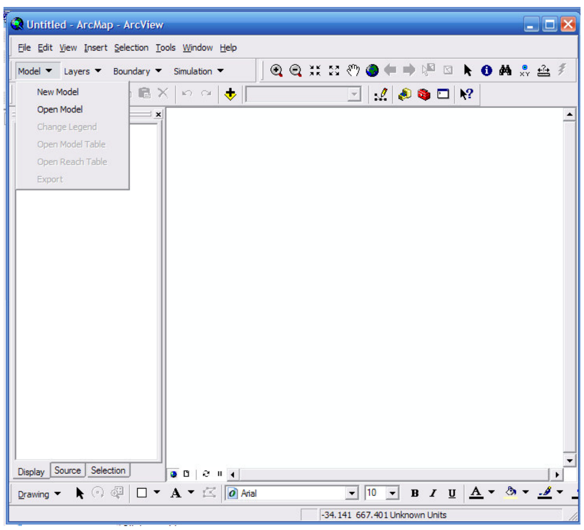
The time series file for providing tidal elevation data at the downstream boundary is: wq_bc.txt. It is an ASCII tab- delimited text file with columns for year, month day, and 8 columns, one for each of the eight advanced quality constituents Details of the required file format and period of record are provided in the User's manual in Section ?.?.

The data in this example file extends from 1/1/1995 to 12/31/2005. Thus, the period of the simulation must be within these limits. However, other time series input files may further limit the simulation period.

Prior to generating a new model, all the configuration GIS files and forcing data text files should be located in a convenient location. For this tutorial, copy the SLE directory from the tutorial CD to the root C: drive. The SLE directory contains a sub-directory called "/configuration_files" which contains all of the input files needed for this tutorial. This folder organization is recommended, but optional, and any folder name and location can be used.

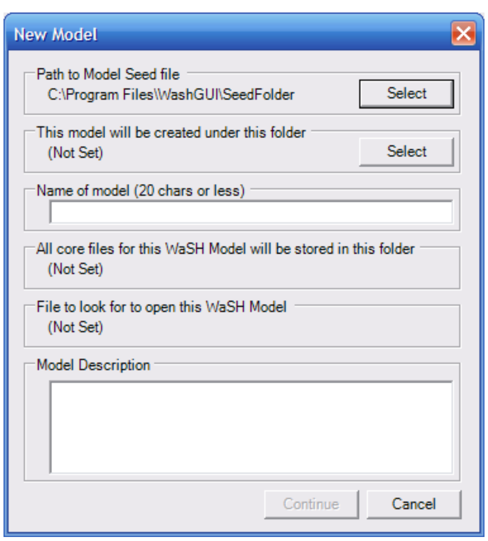
3.1 DEFINING MODEL NAME AND PATH

To create a new model, open an ArcGIS session and initiate the WaSh GUI Toolbar (Section 2.1.3). From the "Model" menu, choose the "New Model" option.



3.1

The New Model screen will allow the user to direct the Seed File path, name a new model, direct the model generation folder and describe the model.

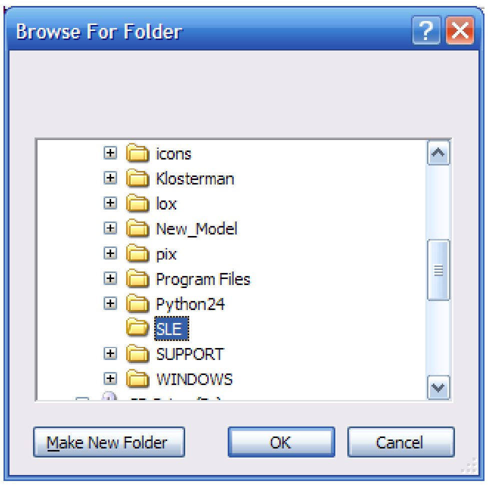


3.2

If this is the first execution of the WaSh GUI, the user will need to specify the Seed File location. The Seed file was placed in a designated path during WaSh Installation. The default file path is: C:\Program Files\WaShGUI\SeedFolder. Use the associated select button, which will launch a browser window. Navigate in the browser to the seed folder and select it. If an alternative path was designated during installation, designate that same path in the browser.

Next, specify the Model folder using the associated select button. Select the SLE directory from the browser. If a project directory was not created prior to starting the model, a new folder can be created within this window using the "Make New Folder" option. Once the folder is created, it is recommended that all configuration and forcing data files be stored in the model directory before continuing with the model configuration.

Additionally, it is recommended that model folders be created on a local or C:\ drive, to increase processing speeds. Operation of the model over Network drives can slow processing speeds and increase model run times.

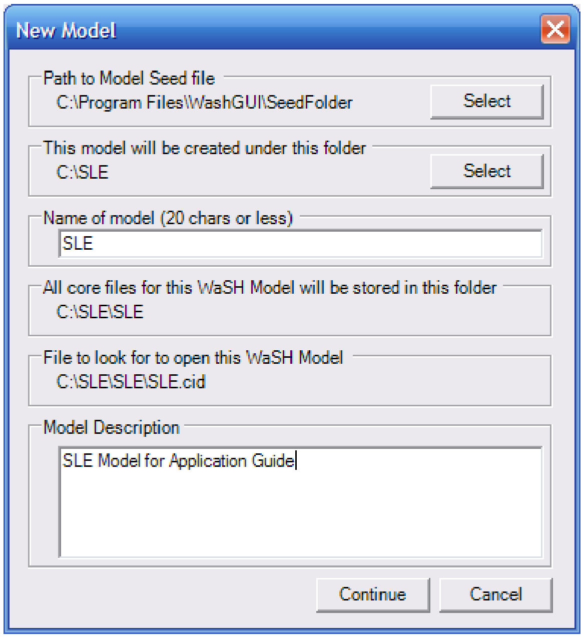


3.3

Next, a name for the model must be provided. Simple model names are suggested, as the original project name is often amended in successive runs created during calibration and model variation. Once the name is defined, the GUI will assign a model core file folder named "SLE" and create a model file called "SLE.cid".

The core file folder will be the location of all of the shapefiles and database files generated by the GUI.

The SLE.cid was created by the GUI using the model name with the ".cid" extension. The model is identified by this file and each time a model is to be reopened, the user will identify the model using the .cid file. If the GUI session is aborted prior to generating the grid (see the next step), then the SLE.cid file will not be valid and the user will have to start with the New Model menu.

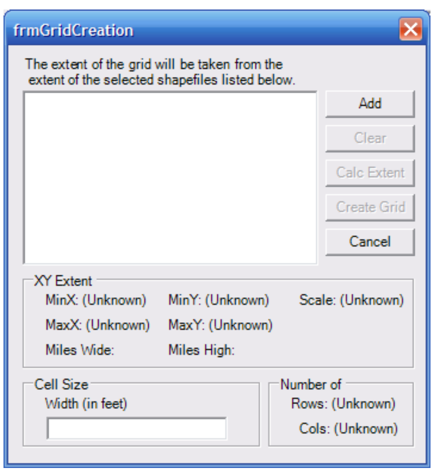


3.4

Type in a model description and select the continue button to proceed to the next step.

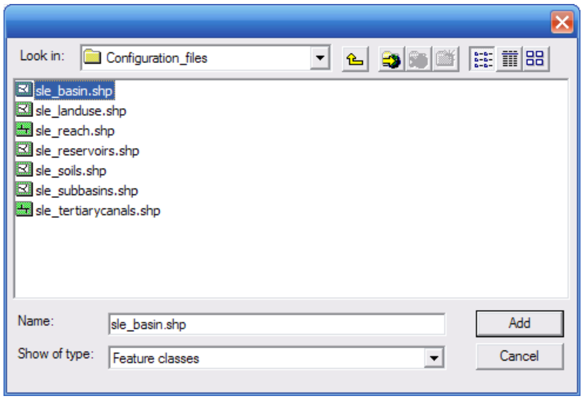
3.2 DEFINING THE MODEL DOMAIN AND GRID GENERATION

The first step in configuring the model is specifying the model domain. The model domain is a rectangle, the extent of which covers the entire basin or basins that comprise the model. The model domain can be defined by the extent of a primary basin, watershed, or river catchment. After having selected the “Continue” button from the “New Model” menu the following menu should be displayed:



3.5

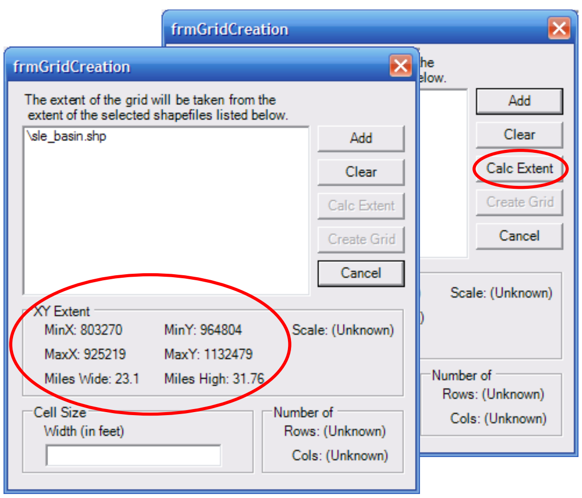
Select the “Add” button, which allows the user to choose the shapefile(s) that will define the maximum extent of the model domain. Navigate to the SLE/configuration_files folder and select the sle_basin.shp file. Then, select the add button.



3.6

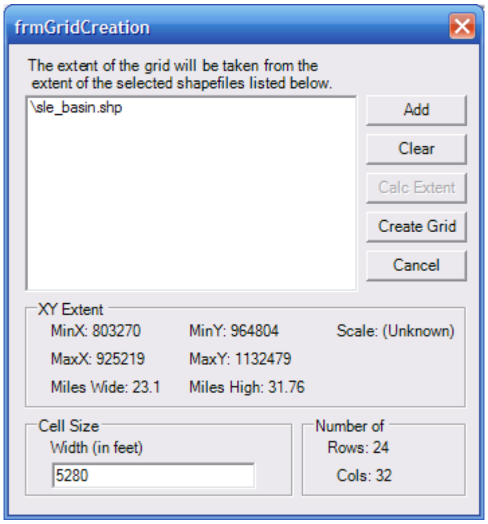
Typically, one file with the basin boundary is chosen. However, by allowing the user to select more than one shapefile, the WaSh GUI can be used to combine several adjacent basins or watersheds into an aggregated model. It is important to note that if multiple files are chosen, they should be adjacent polygons, sharing a common boundary line. As discussed in Section 1.3, the example in this guide uses a model boundary that combined the North Fork Basin, South Fork Basin and Basins 4, 5, & 6 of the SLEW. These files were merged using ArcGIS into a single polygon shapefile prior to model generation.

Now select the “Calc Extent” button. This function will calculate the horizontal and vertical extents of the domain and is useful information for setting the model grid cell size.



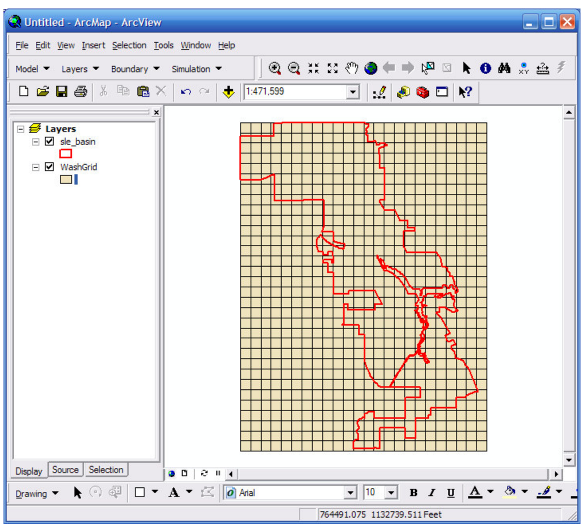
3.7

Next, designate the cell size by typing 5280 into the Cell Size prompt (Figure 3.8). A longer cell length will produce fewer grid cells, lower resolution and faster processing speeds. A shorter cell width will produce more grid cells, finer resolution and slower processing speeds. The GUI will display the number of rows and columns needed to fill the model domain. Select the “Create Grid” button to start the grid generation processing (red arrow).



3.8

Grid generation may take a few minutes, but a timing window will appear to illustrate the generation progress. When the grid has been generated, it will appear as a Cartesian grid as shown in Figure 3.9. In Figure 3.9, the SLE basin boundary has been displayed to show how the grid relates to the specified basin boundary file. The red outline will not be on the user's display. The default coloring of the grid may change with each new model generation, which is an artifact of ArcGIS programming that will not interfere with the WaSh operation.



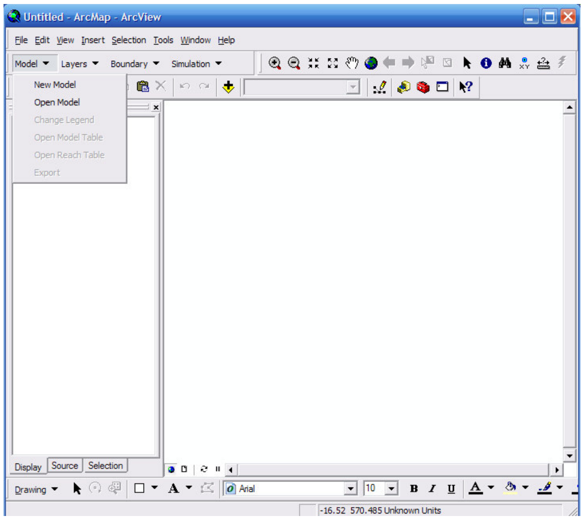
3.9

At the completion of the grid generation process, the SLE.cid file becomes valid and can be used to re-open the model. All subsequent changes to the model are saved instantaneously and automatically by the GUI and there is no need to manually save subsequent work on the model. However, upon exiting ArcGIS the user will be asked if the .mxd file needs to be saved. The .mxd file does not contain any model information and the user should select the 'no' option.

The next section describes the steps for opening an existing model. Close this WaSh model session by selecting the Exit button under the Model menu. This will close the existing wash model and associated files but will leave the ArcGIS running and the WaSh toolbar initiated.

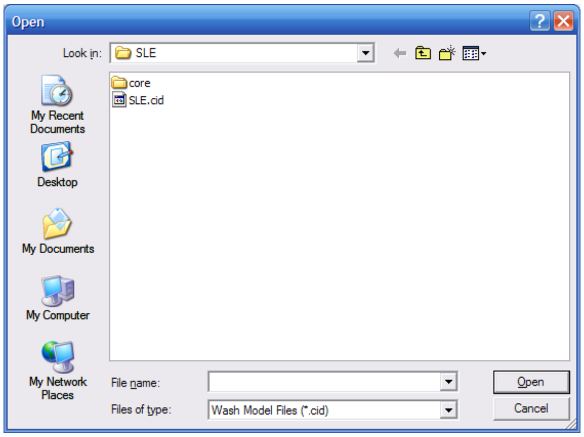
3.3 OPENING AN EXISTING MODEL

Select the Open Model option under the Model menu.



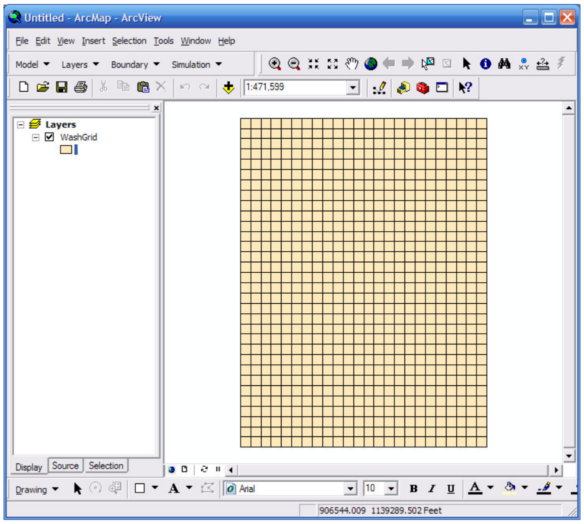
3.10

A windows browser (figure 3.11) will appear. Navigate to the SLE project directory.



3.11

Select the SLE.cid file and then the Open button. The Model will appear as in the figure below. Only the generic Model domain in a monochrome default legend will open initially. If additional model configuration has been completed, the "Change Legend" option in the Model menu can be used to code the grid legend based on the configured basins, land use, soils, or water features.



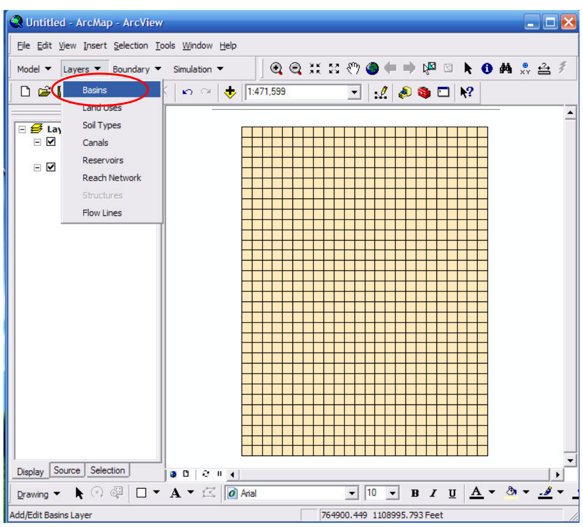
3.12

3.4 ATTRIBUTING THE GRID

The next step in the model configuration is specifying the cell attributes. Tools for mapping physical basin attributes onto the grid are available in the Layers Menu on the WaSh Toolbar.

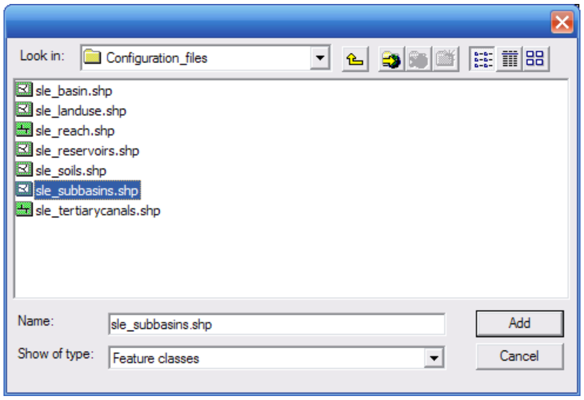
3.4.1 Secondary Basins

Secondary Basins are typically mapped to the grid to help refine the overland flow paths of the watershed system.



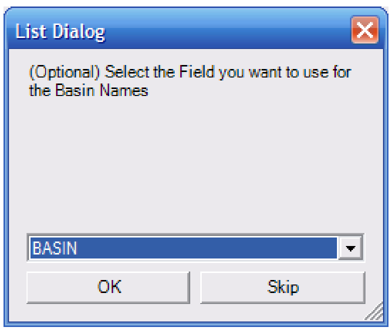
3.13

Select the Basins option from the Layers menu and the following a browser screen will appear. Navigate to the “configuration_file” folder, select the sle_subbasins.shp file, and select the “Add” button.



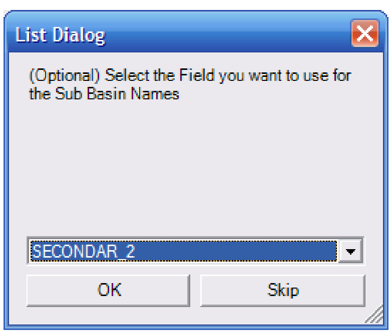
3.14

Once the sub-basin file is added, the user is prompted to specify the Fields within the shapefile that contains the basin name and the sub-basin names. In the basin name query, select the field "BASINS" and then "OK".



3.15

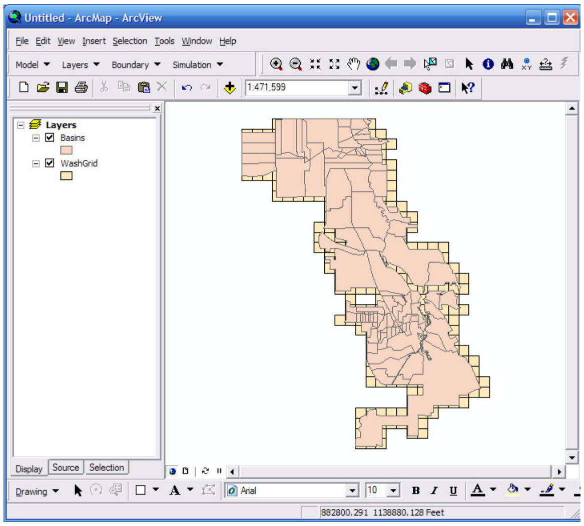
The sub-basin field name menu appears. Select the field name "SECONDAR_2" and then "OK".



3.16

Designating the basin and sub-basin names allows for certain post-processing options to be performed based on the basin or sub-basin designation. The user has the option to skip the field selection process. If this option is chosen, the attribute fields for the grid will be populated with the project name.

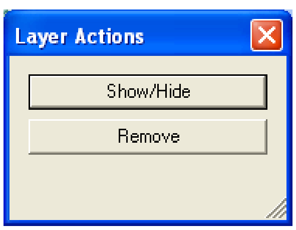
At the completion of this processing step, any cells that are not within the Basin boundary will be considered inactive and are eliminated from the model and no longer shown in the WaSh GUI displays. The GUI displays the active cells in the grid with the original shapefile polygons as an overlay at the end of this step, as shown below:



3.17

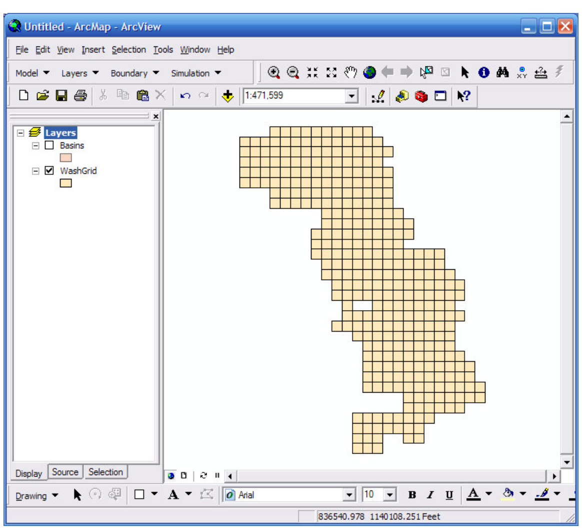
The model grid cells along the edges of the sub-basin polygons often extend beyond the polygon edges due to the irregular basin boundaries. The area assigned to each cell along the edges is reduced in the model to match only the area of the sub-basin polygon coincident with the cell.

When the sub-basin attributing is completed, selecting the Basins options under the Layers menu will launch another menu with additional options. Select the Layers/Basin menu and the following menu will appear:



3.18

Select the "Show/Hide" menu option and the original sub-basin shapefile display will be toggled off and the view appears as shown in Figure 3.17.

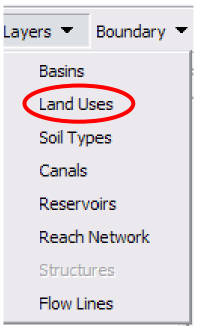


3.19

The Layers/Basin menu shown in Figure 3.18 also has a "remove" option. This option can be used to remove the existing sub-basin mapping and replace it with a new one. At anytime the model grid cells can be color-coded according to existing attributes using the tools described in Section 2.2.2. These general GUI tools apply to the land use, soils and topography mapping procedures also and are described next.

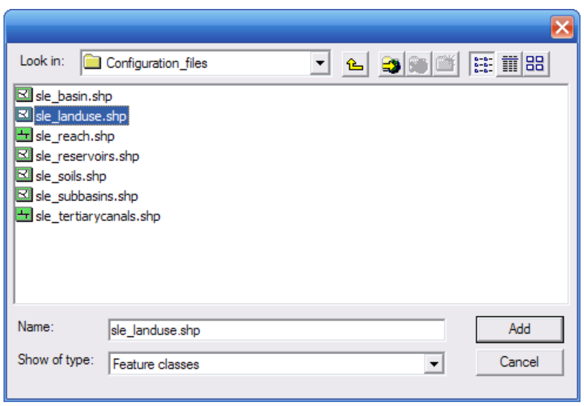
3.4.2 Land Use Mapping

The next option under the Layers menu is Land Use mapping. The WaSh Model is designed to operate using a 3-digit land use code based on the FLUCCS code, discussed in Section 2.3.1.3. To begin the land use mapping, select the "Landuse" option from the Layer menu.



3.20

Using the windows navigator, select the sle_landuse.shp file.

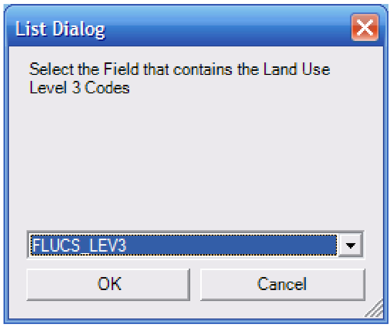


3.21

A land use code level prompt will follow. (see Section 2.3.1.3 for land use code description) Select the Level 3 option.

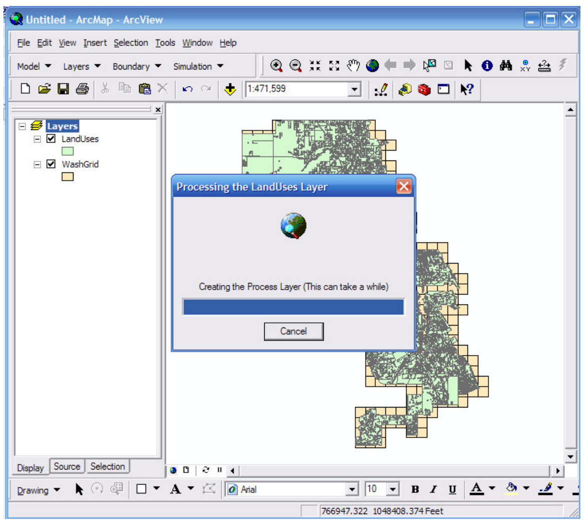
3.22

Next identify the field that contains the numeric land use code by selecting "FLUCS_LEV3".



3.23

Landuse mapping may take several minutes to process, and the GUI display will appear as follows during the processing.

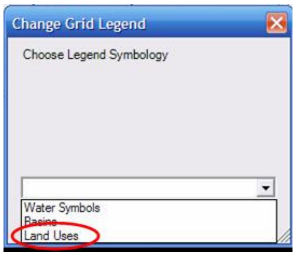


3.24

When the process has completed, the GUI display will contain the active grid cells and the polygons from the sle_landuse.shp file. The land use attribute is now available in the Change legend menu. Remove the sle_landuse.shp file layer by selecting the “Show/Hide” toggle from the Layers/Landuse menu and then change the legend by selecting the “Change Legend” option in the Model menu.

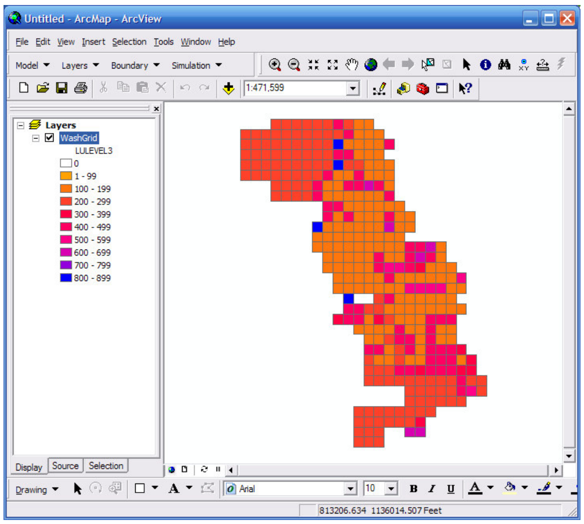
3.25

Choose “Land Uses” from the drop down menu:



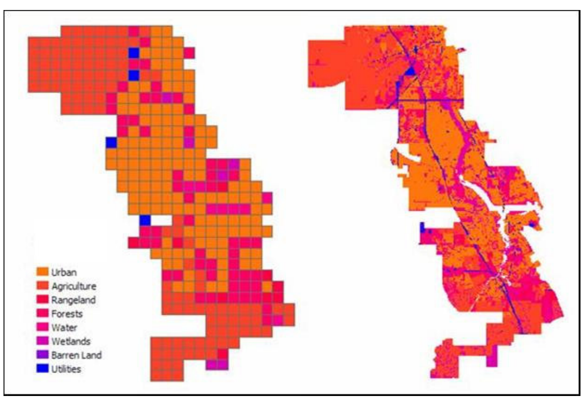
3.26

The Model grid will now be color-coded based on the attributed land use designation.



3.27

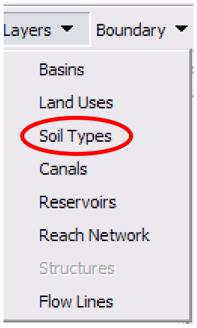
Figure 3.28 shows a comparison of the land use spatial distribution available in the sle_landuse.shp file and the land use as it is mapped onto the model grid. (Note: this figure was prepared for use in this guide and it is not part of the WaSh model GUI).



3.28

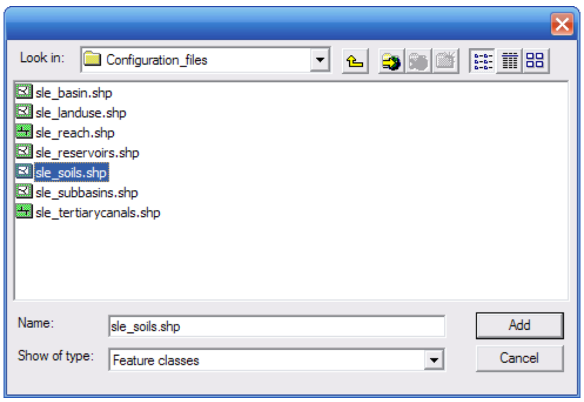
3.4.3 Soils Mapping

Soils attribute mapping is very similar to the landuse mapping. From the Layers menu, choose the Soil Type option.



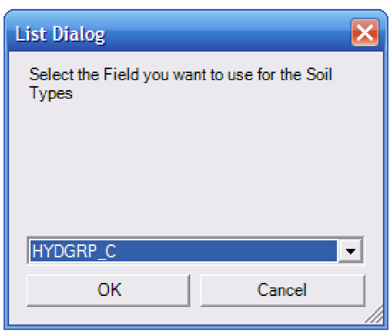
3.29

From the browser window, select the sle_soils.shp file and select the "Add" button.



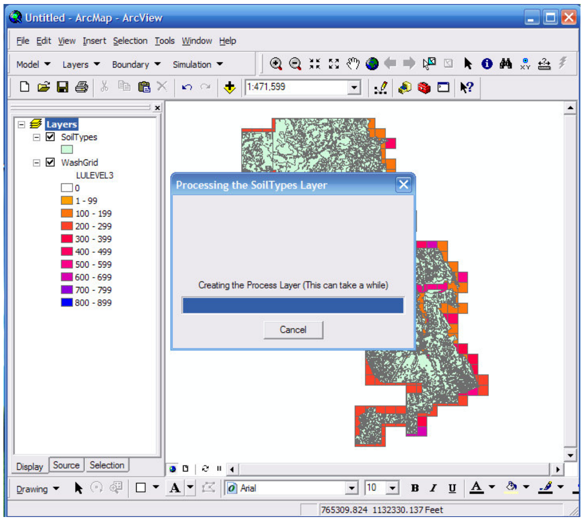
3.30

Next select the field which contains the soils attributes and then select the "OK" button. (More information about soils type attributes can be found in Section 2.3.1.4).



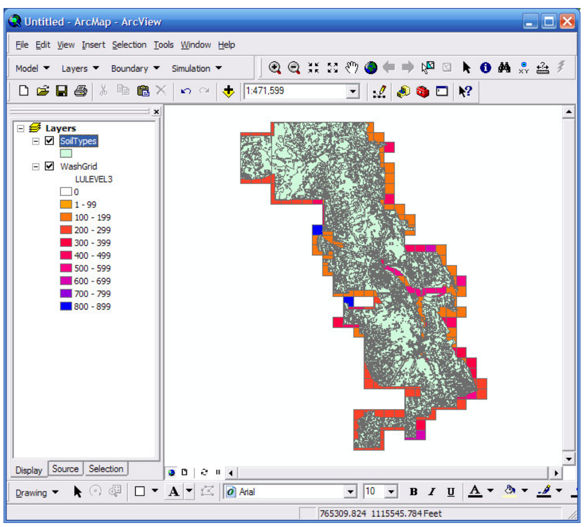
3.31

The model may take a few moments to map the soils attributes onto the model grid and the display will appear as follows:



3.32

When the process has completed, the GUI display will contain the active grid cells and the polygons from the sle_soils.shp file and the cells will still be color-coded according to land use. The land use color coding is shown because it was the last legend chosen (see previous section



3.33

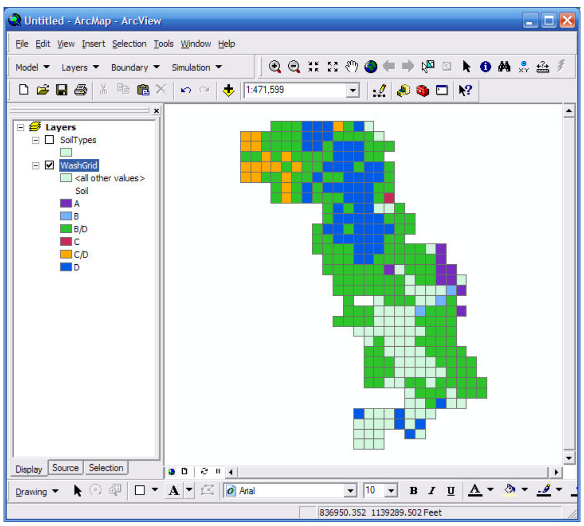
Remove the sle_landuse.shp file layer by selecting the "Show/Hide" toggle from the Layers/Soils menu. The soils attribute is now available in the Change legend menu. Change the legend by selecting the "Change Legend" option in the Model menu.

3.34

Choose the "Soil Types" option:

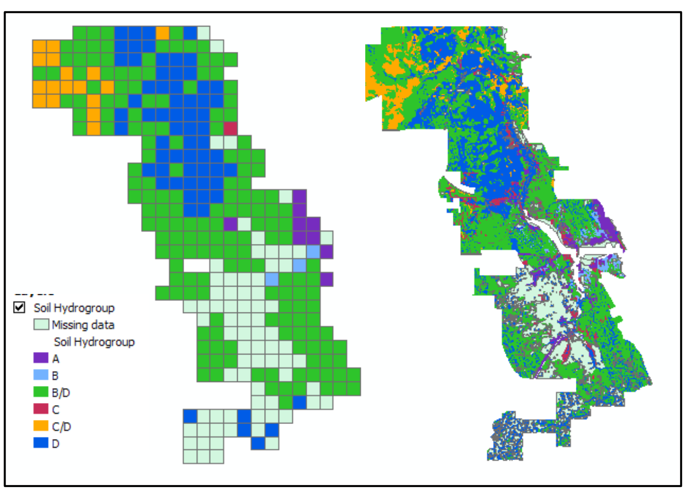
3.35

The wash GUI display will now appear as:



3.36

The following figure illustrates the side-by-side comparison the model grid soil attribute mapping as compared to the configuration soils input file.



3.37

Figure 3.37 shows a comparison of the soils spatial distribution available in the sle_soils.shp file and the soils data as it is mapped onto the model grid. (Note: this figure was prepared for use in this guide and it is not part of the WaSh model GUI).

3.5 DELINEATING SURFACE HYDROLOGY

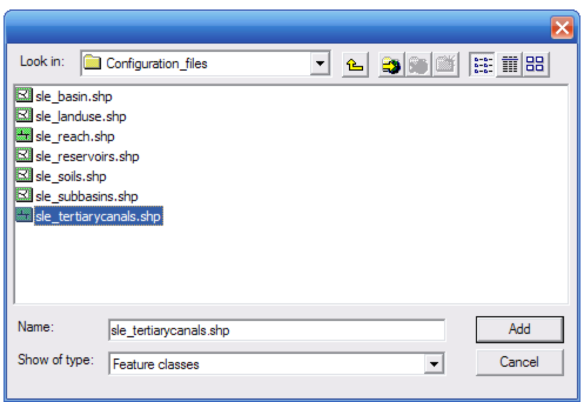
Surface hydrology is comprised of the primary reach and tertiary canals. The primary reach consists of major drainage features, such as named rivers and project canals. The primary reach should be the main drainage flow of the system. The tertiary canals coverage should include tertiary canals, secondary canals, and less prominent natural drainage, such as smaller rivers, creeks, and streams. Both features should be supplied as polyline shapefile coverages. The tertiary canals are actually mapped first, followed by the primary reach.

3.5.1 Tertiary Canal Mapping

From the Layers menu, choose the "Canals" option.

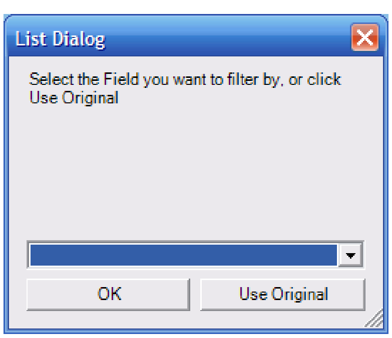
3.38

A file browser will open. Select the sle_tertiarycanals.shp file and select the "Add" button.



3.39

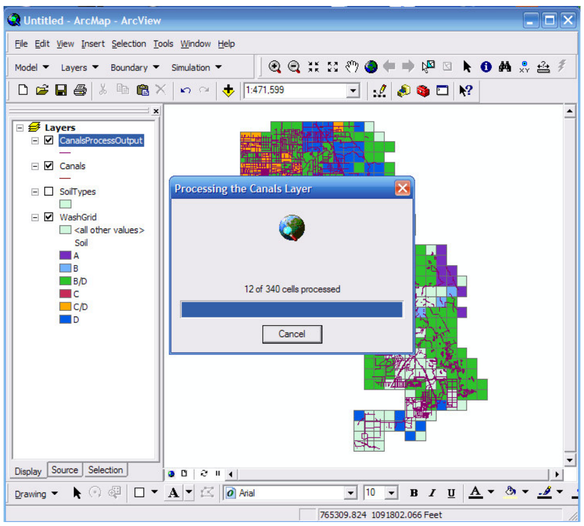
At this point, the user is presented with two options to select the polylines within the shapefile that are used to represent tertiary canals: either "use original" or "filtered".



3.40

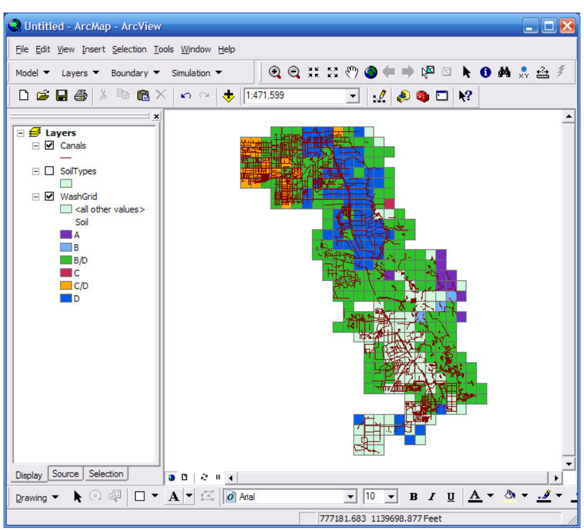
The “use Original” option is the simplest and results in all polylines in the sle_tertiarycanals.shp file to be used. Alternatively, the user could use the “filter” by designating a filter field in the shapefile table. The GUI would then only use those polylines with a non-null field value for the selected field. This application would be useful if the effects of removing a section of the canal system were to be studied. The original file could be amended with a new field and only the tertiary canals that were to remain in the model would contain a non-null vale in the new field.

However, in this example there no such field attribution has been done and all tertiary canals in the sle_tertiarycanals.shp file will be used. Select the “Use Original” button to begin the processing. The mapping process may take several moments during which the display will appear as:



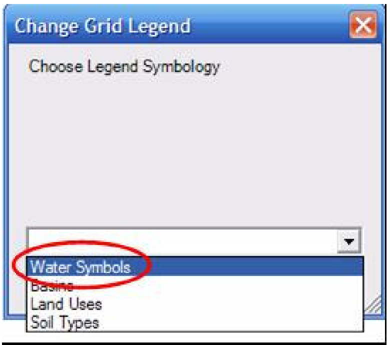
3.41

When the mapping process has completed, the tertiary canal coverage will overlay the model grid and the display will appear with model grid color-coded with the previously selected soils attribute:



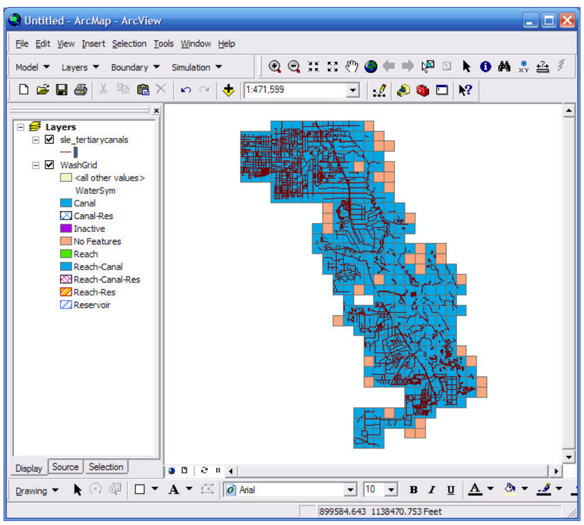
3.42

Choose the “Change Legend” option from the Model menu and select the “Water Features”:



3.43

This will cause the display to show the cell "type" attribute. Two cell types are relevant at this point, "canal" and "no feature". Note that most cells are "canal" cells because of the dense network of tertiary canals in the basin. The grid will appear as:



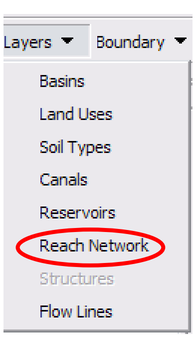
3.44

The polylines from the sle_tertiarycanals.shp feature be removed from the view by selecting the "Show/Hide" toggle button from the Layers/Canals menu.

The cell types will continue to change as other features are added to the SLE WaSh model.

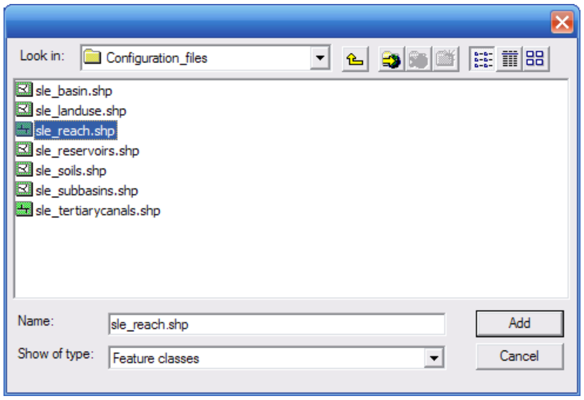
3.5.2 Primary Reach

The second step in mapping surface hydrology is to specify the primary drainage system or the "reach network". From the Layers menu, choose the "Reach Network" option.



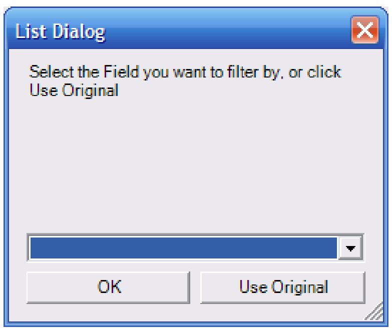
3.45

In the browser screen, select the sle_reach.shp file and select the "Add" button.



3.46

At this point the user is presented with two options to select the polylines within the shapefile that are used to represent the primary reach: either "use original" or "filtered".

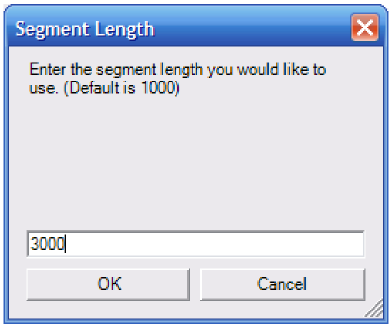


3.47

As described in the previous section, the "use Original" option is the simplest and results in all polylines in the sle_reach.shp file to be used. Alternatively, the user could select a field in the file. The GUI would then only use those polylines with a non-null field value for the selected field. In this example there are no fields available for selecting because the sle_reach.shp file does not have the filter attribute field. Select the "Use Original" button.

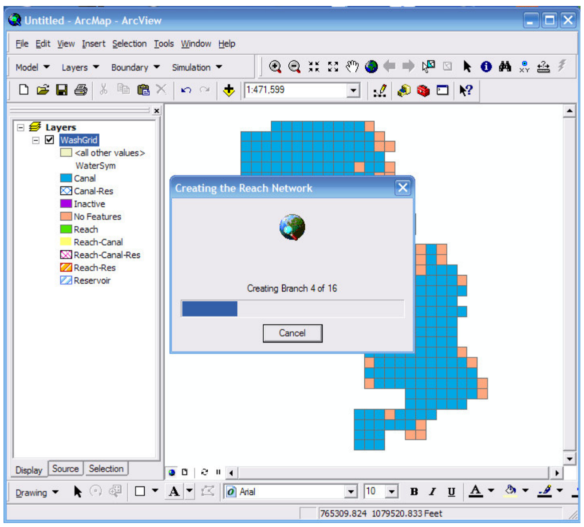
Next, the reach segment length will need to be designated. The segment length must be short enough to provide adequate detail of the flow path of the main channel of the reach but not too short that model simulation time is increased unnecessarily. As with the grid cell size, more reach segments will require additional simulation time. Also, during high flow periods, shorter segments can cause instability in the reach network. Conversely, reach segment lengths should be short enough that the accurate placement of structures and inflow points can be located since the structures must be placed at segment junctions. The default reach segment length is 1000 feet. As a general guideline the segment length should on the same order of magnitude as the cell width.

Set the segment length to 3000 and select the "OK" button.



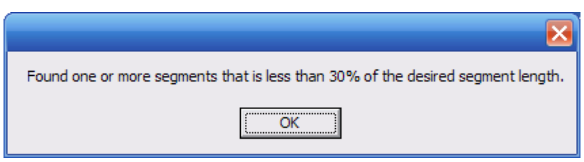
3.48

The reach processing may take several moments during which the display will appear as:



3.49

Occasionally, when larger grid cells are used, the following text box will appear. This is simply an indication that one or more segment lengths were too short and therefore the shorter lengths have been merged automatically, to avoid reach instability issues. Select the "OK" button and the model to continue processing the reach network.



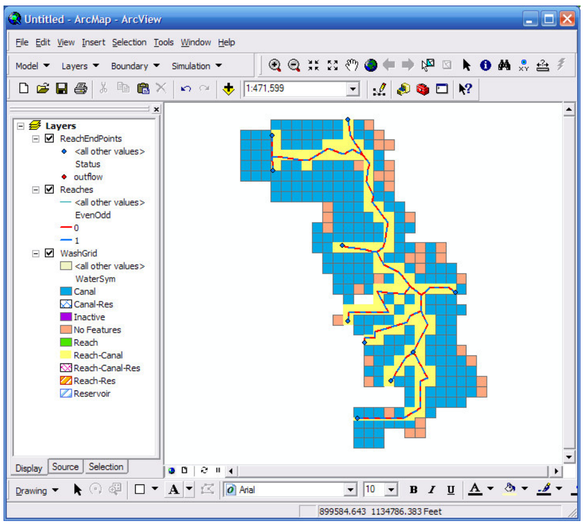
3.50

When the reach network processing is complete, the display will be modified to show (1) the reach network, (2) changes is the cell 'type' attribute and (3) the end points of any branch in the reach network. Figure 3.49 shows these changes.

The reach network appears as alternating red and blue lines. Each alternating color represents a unique reach segment, allowing for the easy identification of reach segment endpoints and center points. Easy identification of the end points is important when adding structures (Section 5.2). In the Reach legend, the red and blue lines are identified with a '0' and a "1". This is purely a segment attribute convention to allow for alternating the color of adjacent segments.

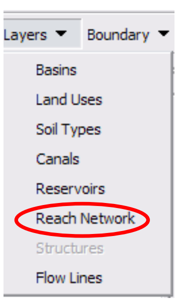
The grid coloring has also been updated to indicate the identification of some cells as "reach-canal" cells. The 'reach-canal' cells now appear with yellow shading. No green cells ('reach' only) exist in this example SLE model, do to the overwhelming concentration of tertiary canal cells.

The "Reach Endpoints" coverage will also be displayed, indicating the terminal points of each reach branch. Initially all endpoints will be colored blue. The next step in the processing is to designate on one reach network terminus as the basin outfall and then perform the flow analysis, which determines the downstream path of the system and numbers the reach segments. A system may have more than one basin outfall point if it contains more than one primary reach. Each basin outflow will need to be designated.



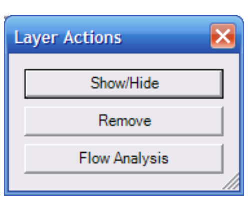
3.51

To designate the basin outfall, return to the Layers menu and chose the Reach Network option again.



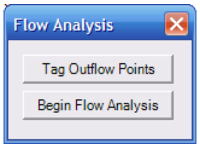
3.52

Since the reach networked has now been created, the menu will provide additional options.



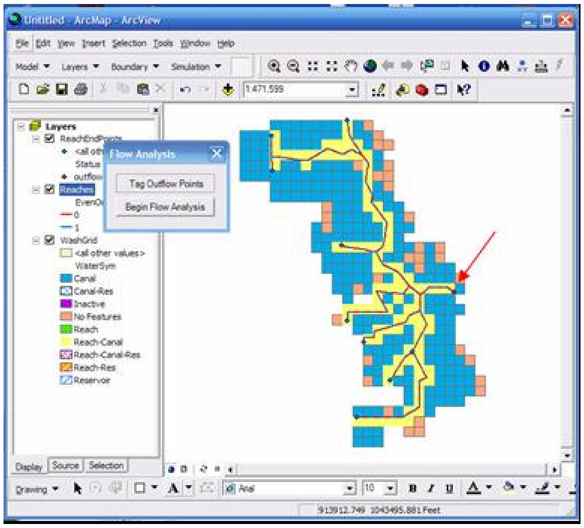
3.53

The "Show/hide" button will toggle the reach network display on and off. The "remove" button allows the user to delete the existing reach network and develop another one. The last button is a tool to designate the basin outfall, which is the last step in creating the reach network. Select the "Flow Analysis" button and the following screen will appear:



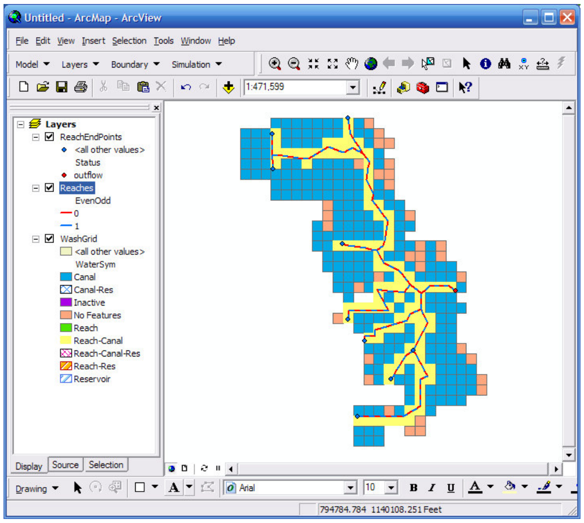
3.54

Select the “Tag Outflow Points” option. The cursor will change in the ArcGIS session, which allows the user to select a terminus point. Select the point as indicated in Figure 3.55.



3.55

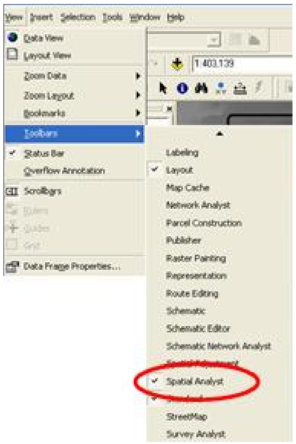
Now select the “Flow Analysis Tool” from the “Flow Analysis” menu. The segments will be numbered and the display will be modified to show the designated basin outfall which is now red, as shown below (pink circle).



3.56

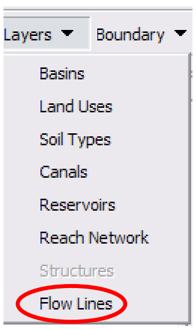
3.6 MAPPING TOPOGRAPHY AND REDIRECTING FLOW PATHS

Mapping topography on to the model grid requires the use of ArcGIS Spatial Analyst Extension. To initialize the Spatial Analyst extension in ArcGIS, navigate to the View menu and select the Toolbars menu. Scroll toward the bottom of the list to see the Spatial Analyst Toolbar and select the option to activate the extension.



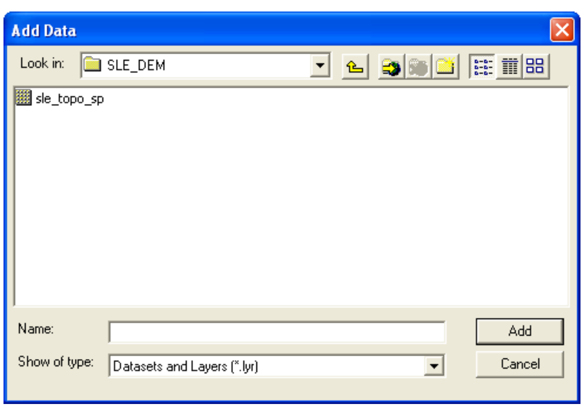
3.57

With Spatial Analyst activated, return to the WaSh Toolbar, select the Layers menu and then select the "Flow Lines" option.



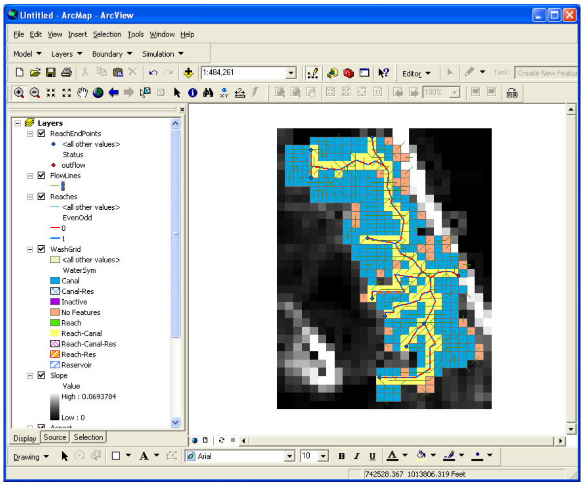
3.58

A browser screen will appear requesting the location of the digital elevation map (DEM) or raster dataset. For more information on DEM's, refer to Section 2.3.1.7. Select the SLE_topo_sp file from the SLE_DEM folder and the select the "Add" button.



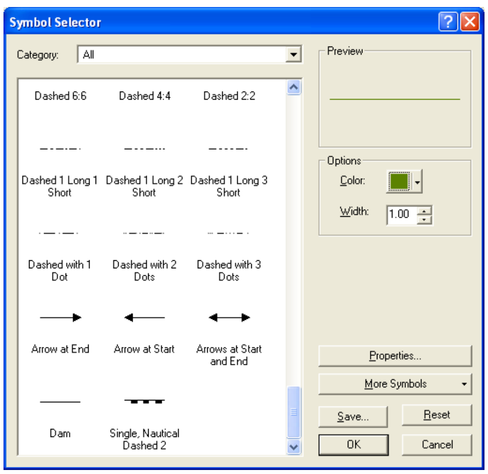
3.59

The GUI will assign the elevations in the DEM to the model grid cells and specify the surface water drainage paths, referred to as 'Flowlines" using topographic gradients based on the cell elevations. The GUI may take a few moments to generate the flow lines. When processing is complete, a line segment will be drawn from the center of each cell to the center of the downgradient cell. The ArcGIS screen will appear as:



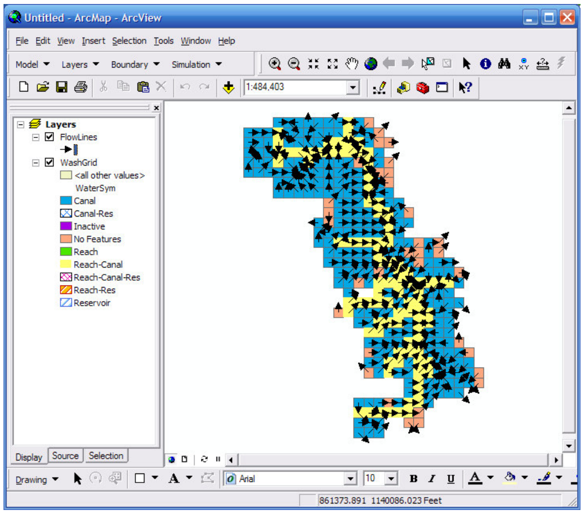
3.60

Next change the “flowline” display from a line to an arrow so that the direction is indicated on the display. To do this, right click on the “Flowlines” in the display legend. Select the “properties” menu, and then select the “symbology” option. From the symbol selector menu, choose the “Arrow at End” option and select the “OK” button.



3.61

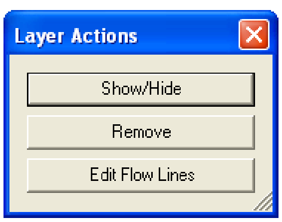
The flow paths will appear as shown in Figure 3.60. The arrows indicate the direction to which surface water flow is directed.



3.62

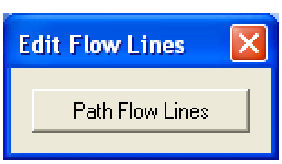
A visual inspection of the flowlines reveals that some flowlines are directing surface runoff out of the model domain, with no receiving cell. In this example, these occurrences can be due to the characteristically flat terrain of South Florida, clipping effects, or boundary processing errors. It is necessary that these errant flowlines be corrected. It may also be desirable to modify flowlines to better reflect known drainage patterns in the basin. Features such as roads and levees that are not reflected in cell elevation may have an impact on the surface flow drainage pattern. Correcting the errant flowlines and any other modification can be done manually using the “Edit Flow Line” tool.

To access the “Edit Flow Line” tool, select Layers/Flowlines and the following menu will appear:



3.63

Select the “Edit Flow Lines” option and when the following menu appears, select the “Path Flow Lines”

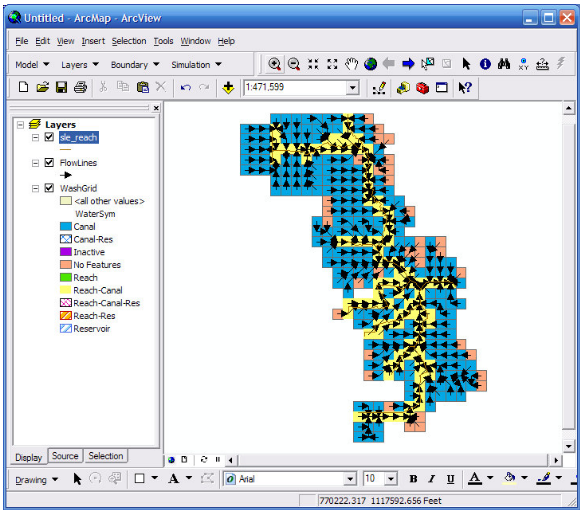


3.64

The “Flow Line Editing” tool will now be active. Flow lines can be edited one line at a time or as a string. To edit a single cell, click on the origin cell (arrow tail) and then double click on the desired destination cell. The flow line arrow will be aligned in the chosen path upon the second mouse click. To edit a string of flow lines, select the origin cell and then click from cell to cell in the along the flow path. With each successive cell selection (i.e. click) the flow line arrow will orient with the chosen path. To end a string, double click in the final destination cell.

The model grid legend is currently set to the “Water Features” option which is typically the view most useful for modifying the flow lines. Other views that may be helpful in some applications are the sub-basin boundaries and the tertiary canals. These can be added using the “Show/Hide” toggles under each features menu in the Layers menu.

For each of the boundary cells that are directed out of the domain, use the flow line edit tool to redirect the flow lines into the domain. Use the figure below as a guide for redirecting the flow lines. When completed the user’s view should be identical to that shown in Figure 3.65.



3.65

Before proceeding to the next step, it is beneficial to remove the flow lines for the display so that other features can be easily located. Go to the Layers/Flow Lines menu and select the “Show/hide” toggle.

3.7 SPECIFYING BOUNDARY CONDITIONS AND TRANSPORT OPTIONS

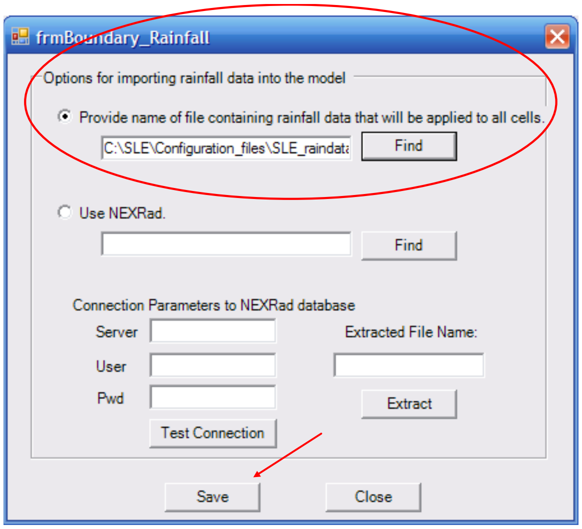
At this point the model is configured with all of the required physical parameters. The final step necessary before launching a simulation is the specification of boundary conditions and model forcing. There are three boundary/forcing conditions that are required for a simulation. The three required conditions are:

- Rainfall
- Potential Evapo-transiration
- Reach network hydrodynamic boundary condition

3.7.1 Rainfall

The WaSh model offers two options for specifying rainfall data. Data can be provided in a time series data file which introduces a uniform distribution of rainfall throughout the basin or as NexRad Doppler rainfall data which provides spatially varying time series of rainfall data. The implementation of the NexRad data requires a direct connection to the SFWMD NexRad database.

A single rainfall time series file will be used in this tutorial. For this example, a time series data file will be used. Select the Rainfall option under the Boundary menu and the following menu will appear:

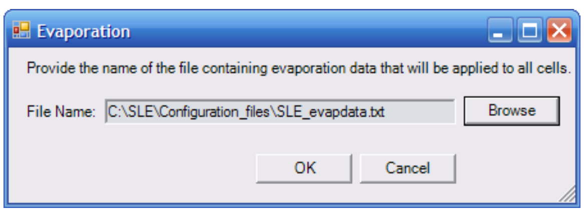


3.66

Select the first option, and then select the "Find" button. Use the navigation window to find the SLE/configuration_files folder and select the SLE_rainfall.txt file. Next, select the Save button to exit the Rainfall menu.

3.7.2 Potential Evapo-Transpiration

The potential evapo-transpiration data is provided as a single time series input file. Select the Evaporation option under the Boundary menu and the following menu will appear:



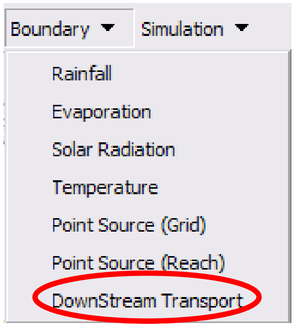
3.67

Select the "Browse" button and use the browser screen that appears to navigate to the SLE/configuration_file folder. Select the SLE_evapdata.txt file. Then select the "OK" button to exit the evaporation menu.

3.7.3 Hydrodynamic Boundary Condition

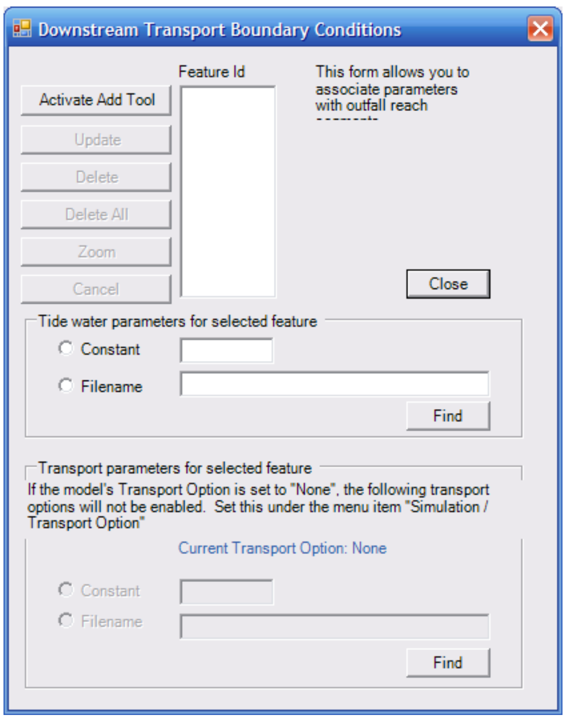
There are two basic hydrodynamic boundary conditions that can be applied: (1) tidal elevation and (2) structural controls. The second option requires the specification of a structure at the downstream segment. This can be accomplished using the "Add a Structure" tool which is discussed in Section 5.0 of this application guide.

For this tutorial, a time-dependant tide elevation is used. The hydrodynamic tide boundary conditions are set in the "Downstream Transport" option from the Boundary menu. Select the "DownStream Transport" menu as indicated in Figure 3.68 and the "Downstream Transport Boundary Conditions" menu will appear as shown in Figure 3.69.



3.68

The general approach for wetting tide boundary conditions is a two-step process. First the source of the tide data is identified (either a constant value or a time-series file) and then the data is assigned to the boundary segment. This two-step process can be repeated for each tide boundary condition that is applied.



3.69

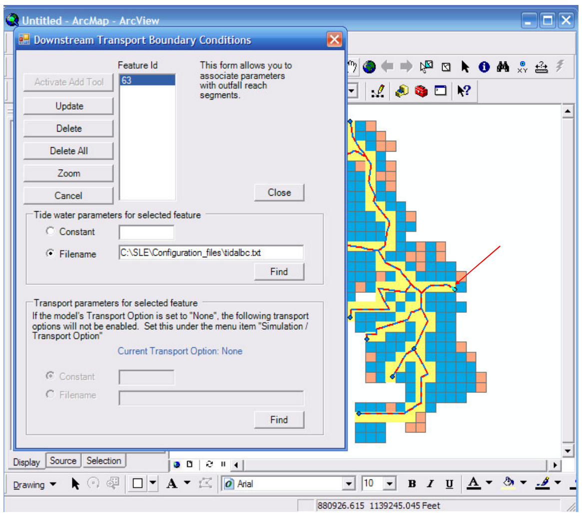
Select the filename option under the Tide water parameters for selected feature", then select the "Find" button to open a windows browser. Navigate to the SLE/configuration_files folder and select the "tidalbc.txt file". Then select the "Add" button.

Next select the "Activate Add Tool" option. The button name will change to read "Activate Tool Enabled". The cursor will be activated in the ArcGIS window, allowing the user to select a reach segment. Select the basin outfall segment (see Figure 3.69 for its location)

If an attempt is made to select a reach segment, but the GUI does not sense that the tool is clicked close enough to the desired reach segment, the following error message will appear.

3.70

Use the Zoom tool in the menu to zoom in on the desired reach segment and repeat the process. When reach selection has been performed successfully, the Arc session will appear as in the figure below.



3.71

Initially, the boundary condition filename will disappear and the reach segment ID number will appear in the Feature ID box. Select the Feature ID number and the associated tidal BC will re-appear. The reach segment will also be highlighted in the Arc GIS session.

Another tidal BC could be added by selecting the "Clear" button, which clears the display but not any previously assigned boundary conditions.

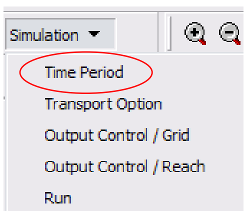
Before exiting the Boundary Conditions menu, note that the last section of the menu is 'grayed out' and there is blue text reading: "Current Transport Option: none". The transport option "none" is the default condition. The transport options here are only necessary when water quality simulation is included with the hydrologic and hydrodynamic simulation. At this point in the tutorial, only a hydrologic and hydrodynamic simulation is being used and the "none" is therefore the correct option. Adding a water quality simulation and the associated boundary conditions is addressed later in this tutorial, in Section 7.0.

Select the "Close" button the menu to exit the DownStream Transport menu.

At this point, the model is configured for a hydrologic and hydrodynamic simulation and the boundary conditions have been established. This section will illustrate how to set the time period of simulation, determine and set output files, launch a simulation and how to perform some basic post-processing of the output data.

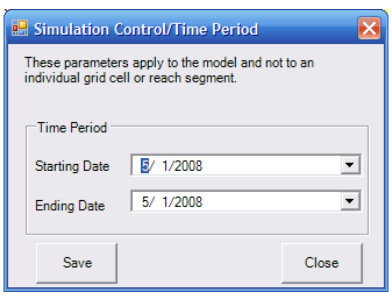
4.1 TIME PERIOD

The simulation time period is accessed through the "Simulation" menu. Select the "Time Period" button.



4.1

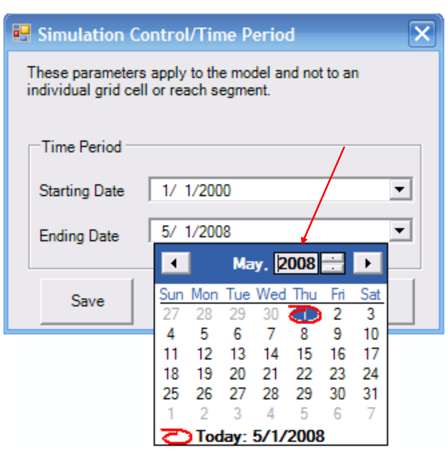
The "Simulation Control/Time Period" screen will appear, allowing input of the beginning and ending dates for the simulation period. For this tutorial, a six-year period will be simulated from 1/1/2000 to 12/31/2005.



4.2

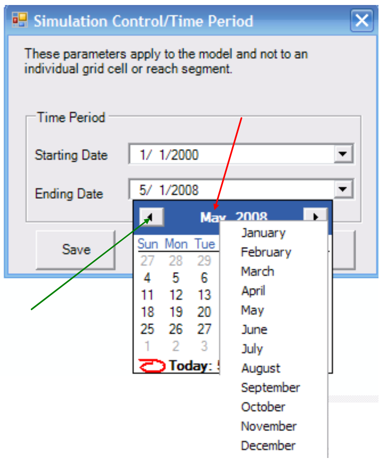
Dates can be entered manually in the provided format (d/m/yyyy) or by using the calendar that can be accessed by clicking the drop-down menu arrow at the right of each date.

Highlight the Starting Date month and enter 1, and then the day and enter 1 and finally the year and enter 2000. The Ending date will be changed by using the drop-down calendar. Select the arrow in the right-hand side of the Ending Date box and the dropdown calendar will appear.



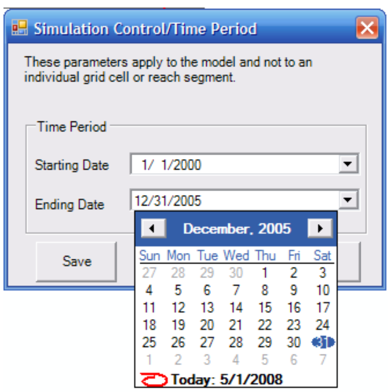
4.3

Click on the year and type in "2005". Double click on the month. A drop down menu appears. Select "December". As an alternative, the scroll arrows on either side of the date (green arrow) allow the user to scroll through month-to-month screens.



4.4

Next, select the 31st day from the December calendar.



4.5

Select the save button.

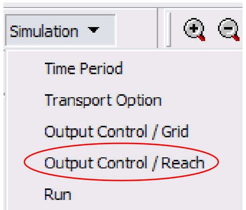
4.2 SELECT OUTPUTS

Model output can be generated for either a reach segment or a grid cell. The basin outfall (reach output) is traditionally the main output studied. However, any output location within the system can be chosen. This function is useful if a calibration station is located upstream of the basin outfall or when a groundwater sampling station or tertiary canal station has available calibration data. Typical output locations are industrial discharge stations, ecological study sites, or specific land use areas.

4.2.1 Reach Output

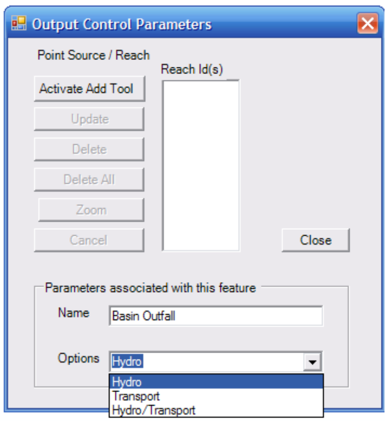
For the SLE model, two reach output locations will be chosen. The basin outfall will be used for quantifying total basin discharge and a segment in the North Fork of the St. Lucie River corresponding to the Gordy Road Station will be chosen, since available calibration flow data is available at that site.

From the Simulation menu, choose the "Output Control/Reach" option.



4.6

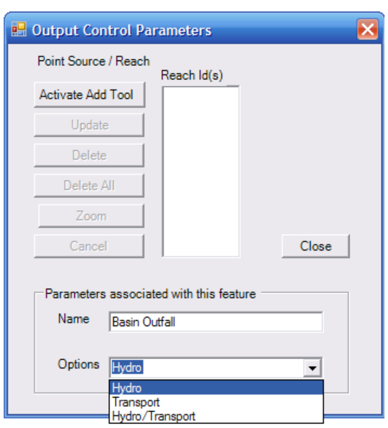
The following screen will appear.



4.7

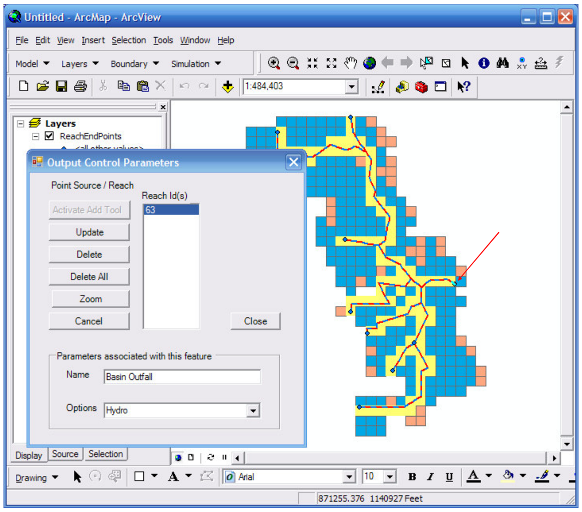
First, designate the output location Name (green arrow). Type in the name "Basin Outfall". Next specify the output option. Select the drop-down menu arrow at the right side of the "Options" dialogue. "Hydro" is the only available option since no water quality simulation has been specified (Adding a water quality simulation is discussed in Section 8).

Next, select the "Activate Add Tool" option. The button name will change to read "Activate Tool Enabled". The cursor will be activated in the ArcGIS window, allowing the user to select a reach segment.



4.8

Select the basin outfall point indicated in the figure below.

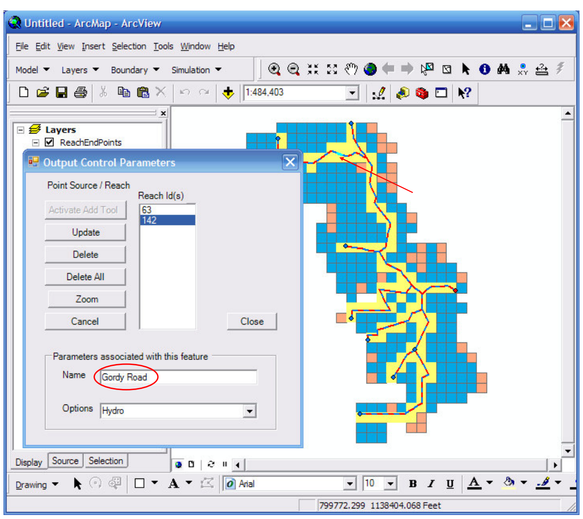


4.9

Upon selecting the segment, the boundary condition filename will disappear, the reach segment ID number will appear in the Reach ID box and the button name will return to "Activate Add Tool". Select the Reach ID number and the associated Tidal BC will re-appear. The reach segment will also be highlighted in the Arc GIS session. The Reach ID should be 63. If it is not, the intended boundary segment was not selected. Select the "Delete" button to clear the highlighted output segment and start over, being careful to select the proper segment. Use the Zoom tool if needed to help identify the segment.

When the Reach ID "63" was highlighted, the "Activate Add Tool" button became grayed out. In order to specify the second output segment, Select the "Cancel" button, which returns the menu to its initial appearance. Using the same procedure, a reach output location will be added for the Gordy Road Station in the North Fork.

Enter "Gordy Road" in the Feature name box (red arrow in following figure) and select Hydro from the Output option menu. Use the cursor to select the reach segment corresponding to the Gordy Road location (red arrow).



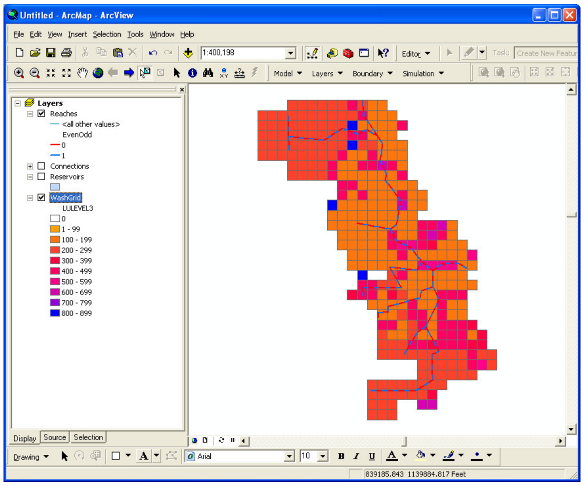
4.10

Select the "Close" button to exit the reach output menu.

4.2.2 Grid Cell Outputs

For the SLE model grid cell output locations, station location choices will be landuse-dependent. Change the legend to "Landuse" using the "Change Legend" option under the Model menu. (Refer to Section 2.2.2).

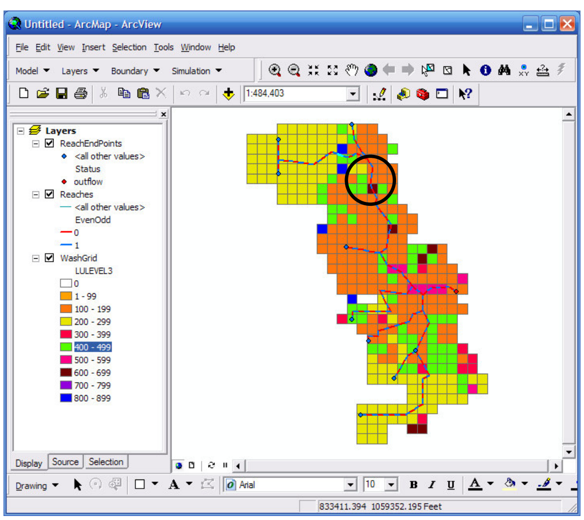
To help identify cells with certain land uses, legend changes should be made manually for agriculture (yellow), wetlands (brown) and forests (green). In the layer list on the left of the ArcGIS session, double click on the WaSh Grid symbol next to 200-299 (Agriculture).



4.11

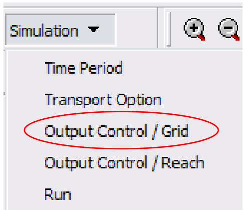
The symbology change menu will open. Change the color to yellow.

Double click on the Forest symbol (400-499) and change the symbol to green. Next choose the "Wetlands" symbol and change the color to brown. The display should now match that in Figure 4.12. Three output locations will be chosen from within the area delineated by the black circle in the following figure. Output from the cells can be compared to study the affects of the landuse on hydrologic output.



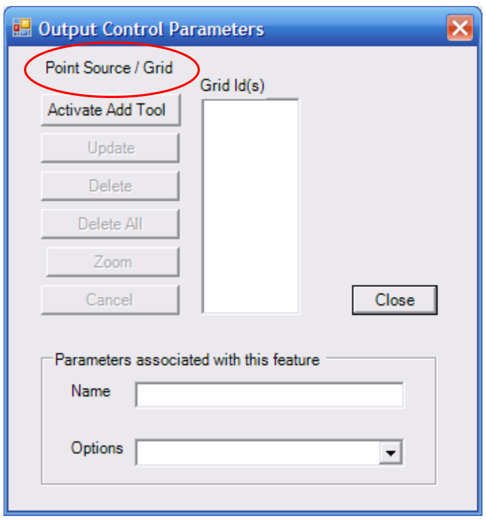
4.12

Select the "Output Control/Grid" option from the Simulation menu.



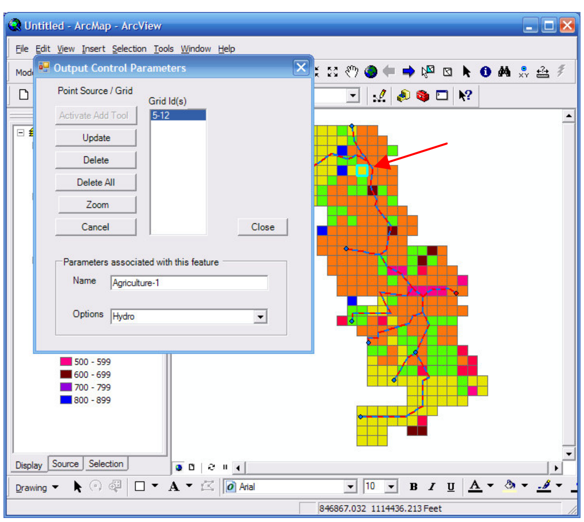
4.13

The following screen will appear.



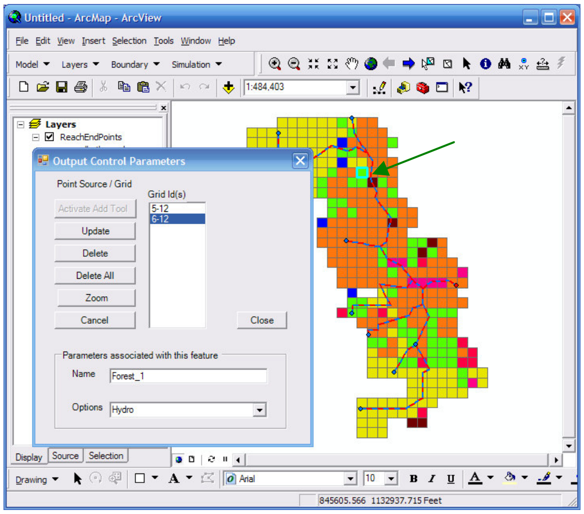
4.14

The process for specifying cells for output is very similar to that for specifying reach segment outputs. Type “Agriculture_1” into the “Name” field, assign the output transport option as Hydro and select with the “Activate Add Tool” button. Use the cursor to select the cell identified by the red arrow in the figure below.



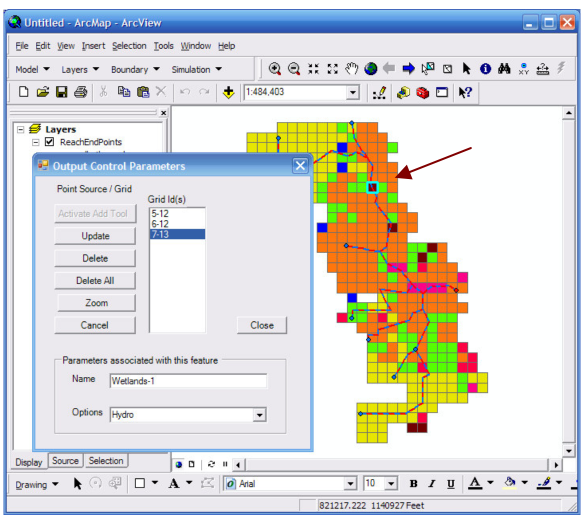
4.15

Now type "Forest_1" into the Name field, select "Hydro" from the Options field, and the select the "Activate Add Button". Choose the forest cell identified by the green arrow in the figure below.



4.16

Finally, to specify the third cell, type "Forest_1" into the Name field, select "Hydro" from the Options field, and the select the "Activate Add Button". Choose the wetlands cell identified by the brown arrow in the figure below.

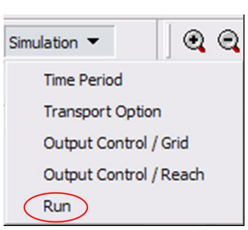


4.17

Select the "Close" button to exit the cell output menu.

4.3 LAUNCH SIMULATION

To initiate a WaSh Model simulation, choose the Run option from the Simulation menu.



4.18

A DOS screen will appear and will continually update the simulation date as the simulation progresses. When the simulation is complete, press Enter on the keyboard and the DOS screen will disappear.

4.4 POST-PROCESSING OUTPUTS

This section is awaiting the final WaSh model outputs

4.4.1 Plotting Reach Output Files

This section is awaiting the final WaSh model outputs

4.4.2 Plotting Cell Hydrology Output

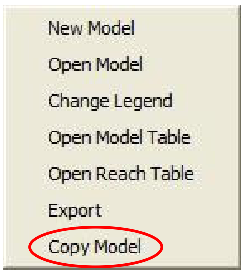
This section is awaiting the final WaSh model outputs

The Gordy Road control structure is located in the North Fork of the St. Lucie River. It is a four-gate spillway, and is operated to maintain a 15-foot water elevation on the upstream side. For the purpose of simulating the impact of this structure on the drainage, it will be represented as a weir with a static 15-foot crest elevation and a 72-foot width.

5.1 CREATING A NEW MODEL FROM AN EXISTING MODEL

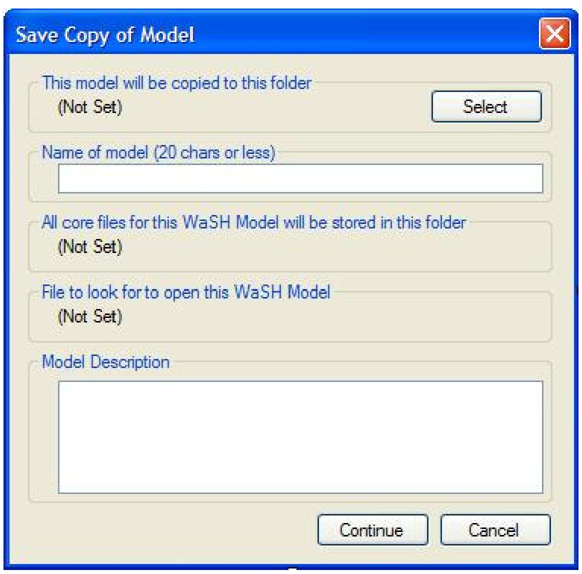
First, a new model will be created for the existing model. This will preserve the output from the previous model. If the structure was added to the existing model and the model simulation executed, then all of the previous output data would be overwritten and lost.

Under the Model menu, select the "Copy Model" option.



5.1

The following screen will appear:



5.2

Using the "select" button, navigate to the SLE folder. In this application, we will store the new model under the same SLE folder as the previous SLE model. Next, type "SLE_Gordy" into the Name of Model field. In the description field, type "SLE model with Gordy Road structure".

Select the "Continue" button. All of the configuration, database, legend and grid files associated with the original SLE model will automatically be copied as part of the "SLE_Gordy model".

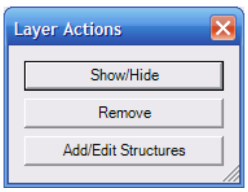
The new model will open and display a default generic color code. Use the Change Legend feature (under the Model menu) to code the grid by Water Features and then select the "Show/Hide" option under the Layers/Reach Network menu. This will display the grid cell type and the Reach Network.

5.2 STRUCTURE MENU

Choose the "Structures" option from the Layers menu.

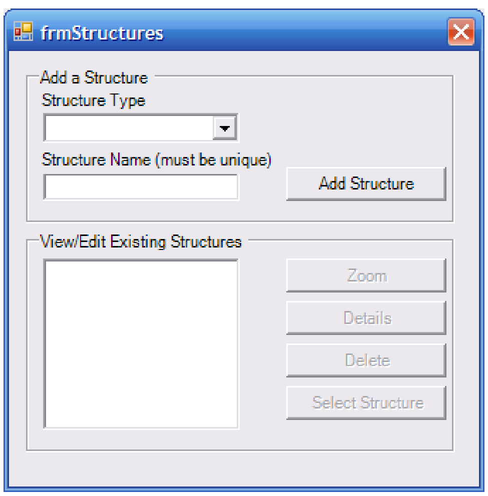
5.3

The following screen will open.



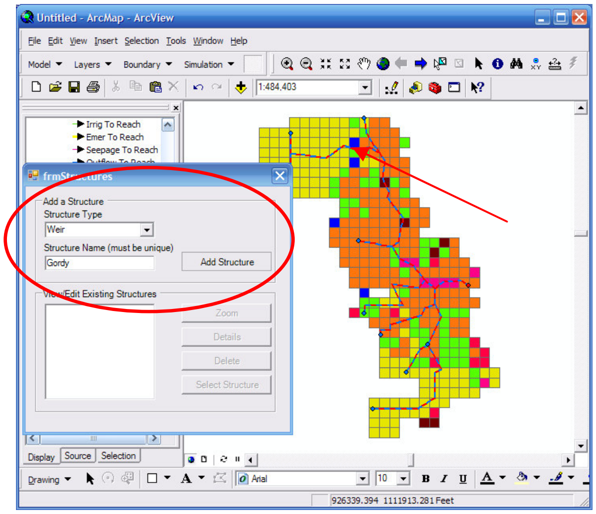
5.4

Select the Add/Edit feature to open the following structures screen.



5.5

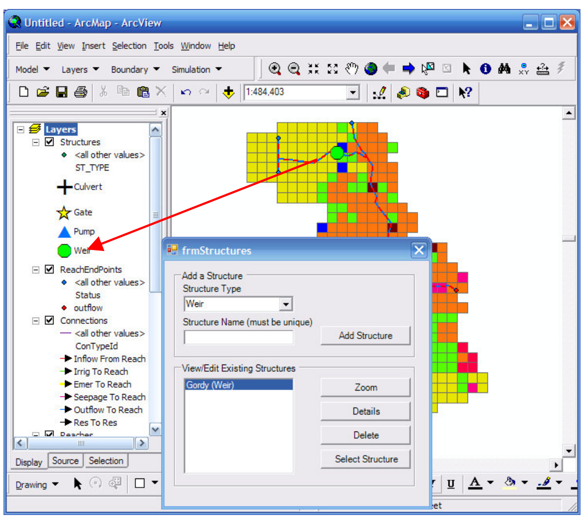
Access the Structure Type pull-down menu and select the "Weir" option. Type "Gordy" into the Structure Name field, then select the "Add Structure" button.



5.6

The cursor will change to a selection tool allowing the structure location to be chosen in the ArcGIS session. Structures must be placed between two reach segments. Use the Zoom feature to help identify the junction of two segments more easily.

Select the structure location with the cursor tool at the location indicated by the red arrow in the above figure. Once selected, a feature symbol will appear at the juncture and also in the Legend, as shown in the figure below.

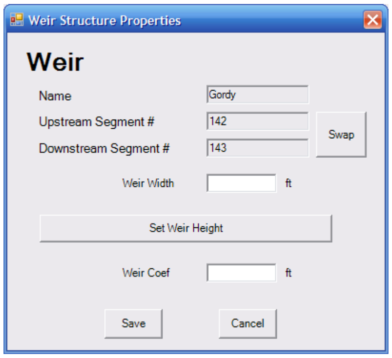


5.7

Now that the structure is located, structure details will need to be added. From the Structures screen, highlight the “Gordy” structure in “View/Edit Existing Structures” window and select the “Details” button. The menu shown in Figure 5.8 will appear:

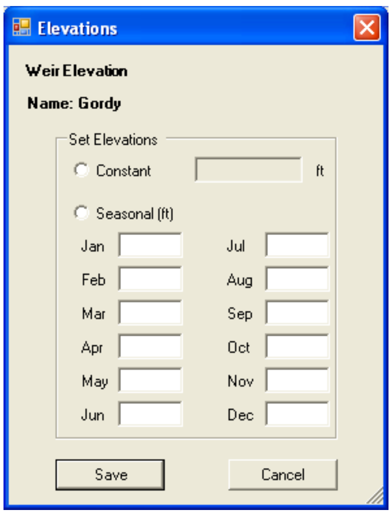
The name “Gordy” will appear in the Name field, indicating which structure has been selected. The next two boxes identify the two reach network segments connected by the structure. They are highlighted on the graphical display also. The upstream/downstream designation is important for certain structures when their operations are dependent on headwater or tailwater conditions. If the upstream/downstream direction is not correct, the user can then “Swap” button to reverse the direction. In this application the use of a weir is not dependent on direction, so the “Upstream Segment” and “Downstream Segments” are left at their default values.

Enter "72" into the Weir Width field.



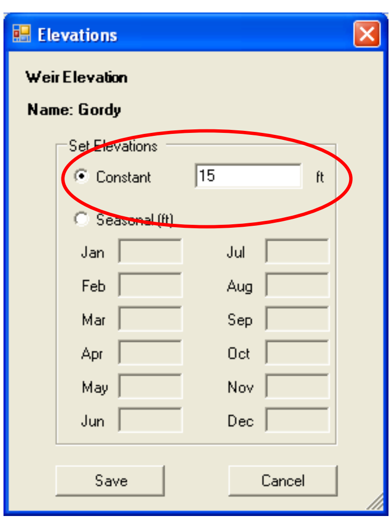
5.8

Select the "Weir Height" button, which will open the following screen.



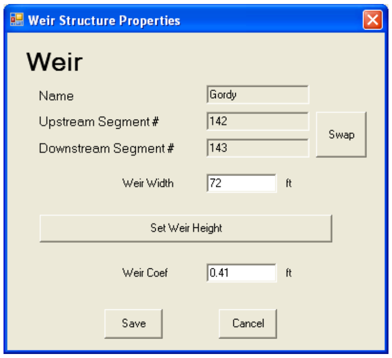
5.9

The menu provides the option to specify a single constant weir elevation or a seasonal (monthly) value. The second option can by used to change the weir to represent different seasonal structure operations. The constant value is used in the tutorial. Select the "Constant" option and type in "15".



5.10

The last structure parameter is the Weir coefficient. This coefficient represents the effect of the weir crest geometry. Further information is available in the WaSh model Theory Manual. Type “0.41” into the ‘”Weir Coef.” field. The completed Structure Properties screen will now appear as in the following figure:



5.11

Select the “Save” button to exit the structures menu.

5.3 LAUNCH AND REVIEW AFFECT OF STRUCTURES ON OUTPUTS.

From the Simulation menu, select “run” to launch the simulation. A DOS screen will appear and will continually update the simulation date as the simulation progresses. When the simulation is complete, press Enter on the keyboard and the DOS screen will disappear.

The remainder of this section is awaiting the final WaSh model outputs

Reservoirs can be added to a model in one of two ways, using a polygon shapefile delineating the reservoir boundary or manually digitizing the extents of the reservoir directly onto the grid. Both methods will be used to add two reservoirs to the SLE_Gordy model.

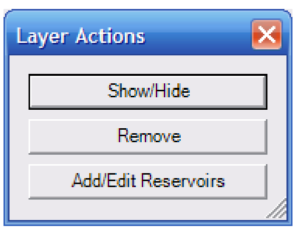
Using the guidelines provided in Section 5, make a new copy of the existing model called "SLE_res".

6.1 THE RESERVOIR MENU

All reservoir functions are accessed through the Reservoir Option on the Layers menu.

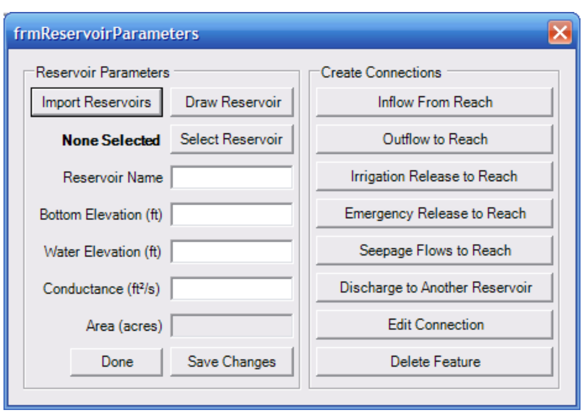
6.1

Select the Reservoirs Option to open the following screen, which provides tools to show or hide, remove, add or edit reservoirs.



6.2

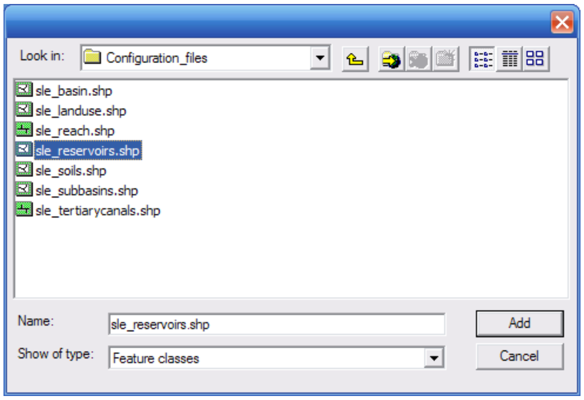
Select the "Add/Edit" option to open the following screen.



6.3

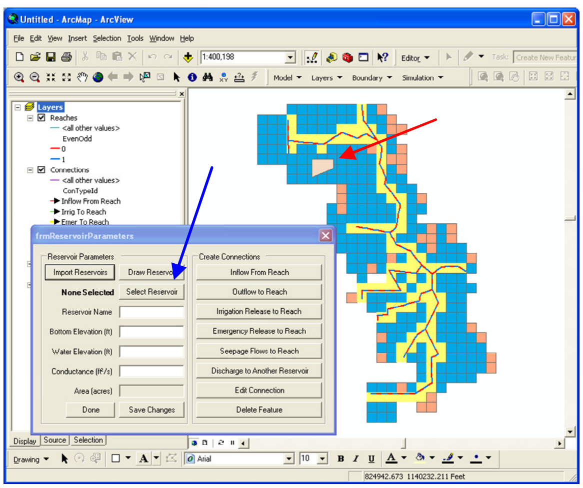
6.2 ADDING A RESERVOIR FROM A SHAPEFILE

First, a reservoir will be added using the provided shapefile, sle_reservoirs.shp in the configuration files folder. Select the "Import Reservoir" button from the previous figure.



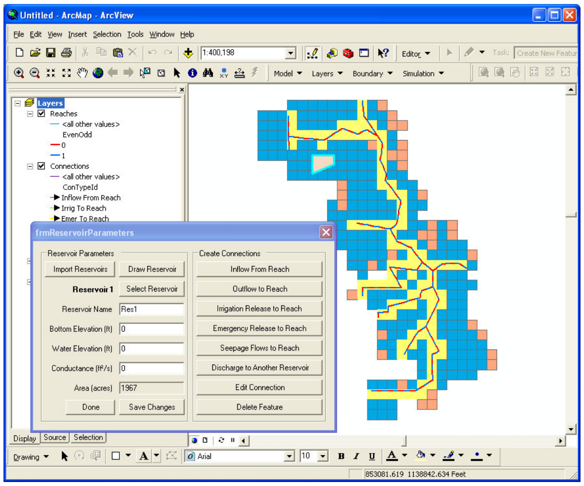
6.4

Use the browser to navigate to the SLE/Configuration_files and select the sle_reservoirs.shp file. The reservoir will appear in the ArcGIS session, as indicated by the red arrow on the following figure.



6.5

Select the "Select Reservoir" button (blue arrow) to activate the selection cursor in the ArcGIS session. Click on the new reservoir polygon.



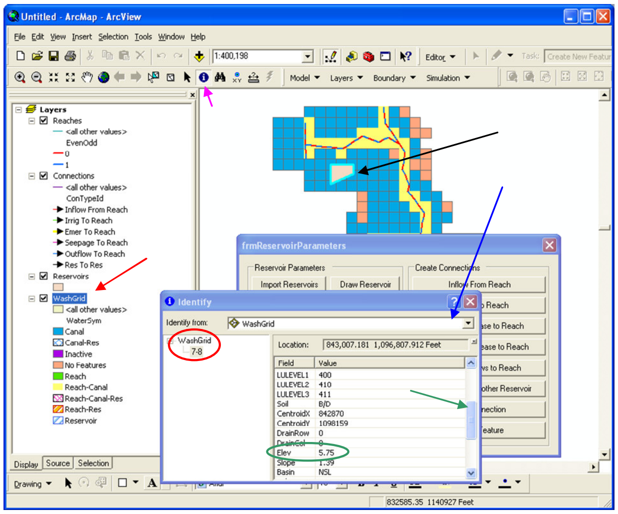
6.6

Selecting the reservoir will add the cyan highlight to the polygon which automatically populates the open fields in the box on the left-hand side of the Reservoir Parameters screen. These are default values and will need to be adjusted to represent the actual reservoir represented in the model. The acres field is automatically populated with the area of the reservoir polygon.

6.3 ASSIGNING RESERVOIR ATTRIBUTES

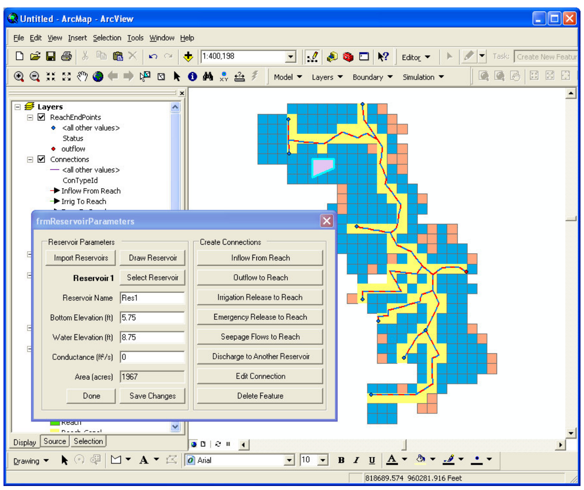
In this tutorial, the default reservoir name "Res1" will be used, although the user can choose any name by typing it into the Name field. Also, assume that the reservoir is perched above grade and lined. Since the reservoir is perched, the bottom elevation is the ground surface elevation.

The next steps are used to identify the ground surface elevation so that it can be entered into the "Bottom Elevation" field. Highlight the WaSh Grid layer in the Layer list (left section of the ArcGIS session at red arrow) and then activate the "Identify" cursor by selecting the icon at the pink arrow. Using the "Identify" cursor, select a grid cell covered by the reservoir (black arrow). The "Identify" Screen will appear showing the attributes of the feature chosen. If the layer chosen is not the WaShGrid (red circle), choose the WaShGrid layer from the pull down menu at the blue arrow. Use the scroll bar at the right of the screen (green arrow) to scroll through the list of the attributes of the Grid cell. The "Elev", or elevation attribute will give the ground surface elevation of that cell, which is 5.75 feet. Select the red "x" in the top right of the Identify screen to close it.



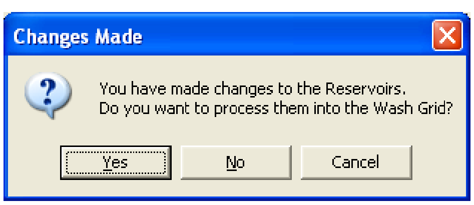
6.7

Type "5.75" into the Bottom Elevation field and "0.0" into the Conductivity field. (***Note to reviewers: Water Elevation Field will be removed from menu – it is not needed).***



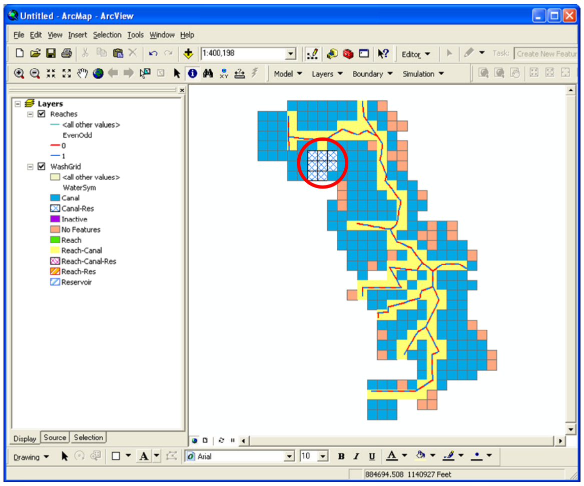
6.8

Select the "Done" button. This will save the reservoir parameters and change the cell types. Grid cells completely under the reservoir will become inactive. Cells partially covered by the reservoir will become "reservoir" cells. The area of these cells will be reduced by the area covered by the reservoir. The following screen will have appeared:



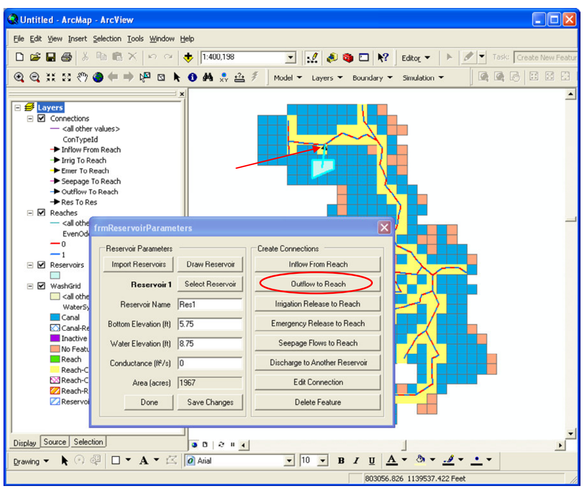
6.9

Select the "Yes" button to initiate the attribution process. After the processing is complete, choose the Reservoirs option from the Layers menu and select the "Show/Hide" button. The grid will now appear as below. The red circle identifies the location of the reservoir. Note the change in the color-coding to the reservoir cells.



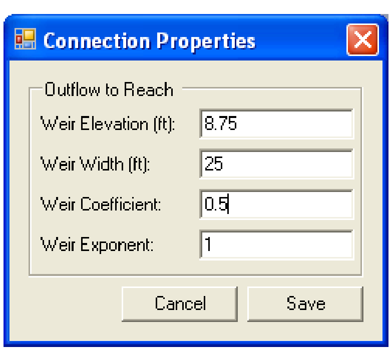
6.10

The reservoir in this example will drain over a weir to a nearby reach. To specify this connection, select the "Res1" reservoir using the "Select Reservoir" tool. Then select the "Outflow to Reach" option (red circle) in the next figure. Move the cursor to the reach segment identified in the next figure (red arrow), and select the segment.



6.11

The Outflow to Reach Connection Properties menu will then appear. Type "8.75" into the "Weir Elevation" field, "25" into the "Weir Width" field, "0.5" into the "Weir Coefficient" field and "1" into the "Weir Exponent" field. The weir crest elevation of 8.75 represents a 3-foot height above the reservoir bottom elevation of 5.75 feet. Thus, the reservoir will have nominal water depth of 3 feet.

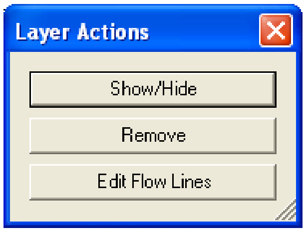


6.12

To exit and save the values, select "Save" button.

The addition of the reservoir may not be consistent with the existing surface water drainage. In this example, the new reservoir will be designed to collect surface water run-off from the surrounding cells. Therefore, the flow paths that were originally routed towards the reach network will need to be rerouted to the reservoir cells. Using the same procedures outlined in Section 3.5, flow paths will be redirected into the reservoir.

Choose the Flow Lines option from the Layers menu and the following menu will appear.

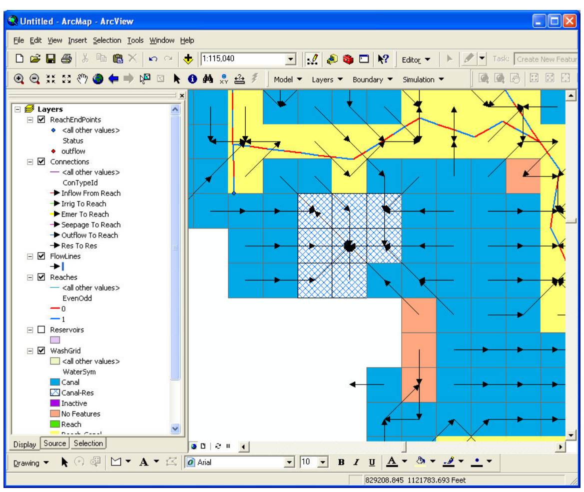


6.13

Selecting the "Edit Flow Lines" option will open the following screen.

6.14

Select the Path Flow Lines option and click on each cell within and adjacent to the reservoir. Redirect the flow inward. When the realignment of the flow lines is complete, select the "Finish" button. The flow lines in the vicinity of the reservoir should appear similar to the following figure:

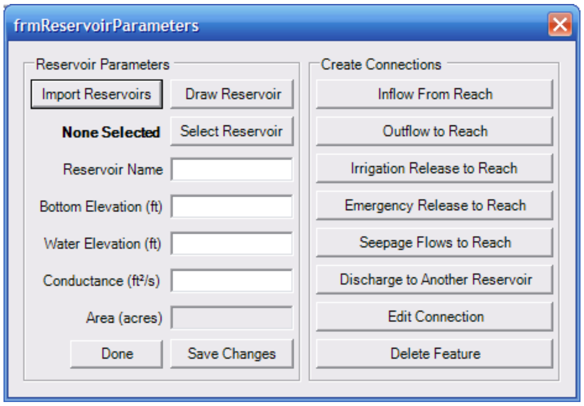


6.15

6.4 DIGITIZING A RESERVOIR

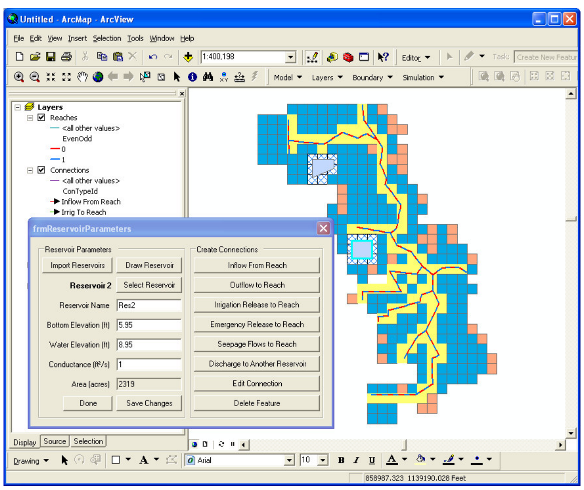
Next, a proposed regional water storage reservoir will be added to the model. The reservoir will not interact directly with the adjacent areas, but will draw water directly from the reach and will discharge in emergency situations. In this case there is no GIS coverage as in the last example to provide the reservoir footprint and therefore, the reservoir footprint will be digitized directly on the model grid.

To digitize a reservoir, choose the “Draw Reservoir” option from the Reservoir Parameter menu.



6.16

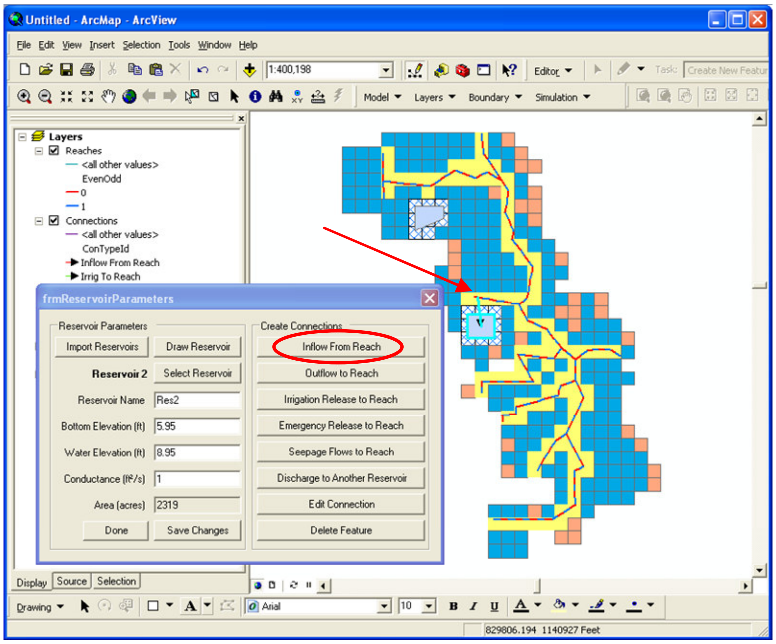
The cursor will be programmed to digitize a polygon. Digitize the polygon as indicated in the next figure, with as many vertices as desired, and double click to close the polygon. Use the “Select Reservoir” tool to select the polygon as was done in Section 6.2.



6.17

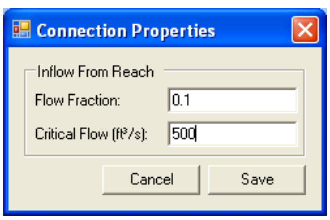
Use the default name. Determine the ground elevation using the same procedure as in Section 6.2. It is 5.95 ft. Then, set the Bottom Elevation to 5.95 and the Conductance to 1 ft^2/day.

To connect the reservoir to the reach network, select the “Inflow from Reach” button from the Reservoir Parameters screen (red circle) and use the cursor to select the reach segment from which the flow will come (red arrow).



6.18

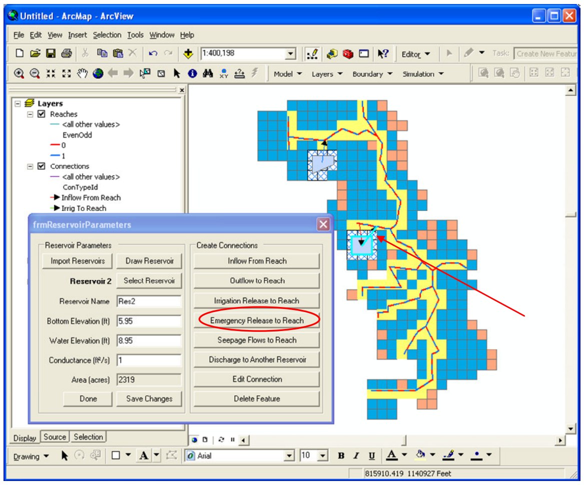
The following screen will appear and prompt the user for the connection parameter values.



6.19

Type “0.1” (i.e. 10%) into the “Flow Fraction” field and “500” cfs into the “Critical Flow” field. These values specify that 10% of all flows above 500 cfs will be extracted from the reach and routed to the reservoir. A more detail description of these operations and connections are available in the WaSh Model User’s Manual and in the Model Theory Documentation. Select the “Done” button and the reservoir will be saved to the model and the Water Symbols legend will be updated in the grid.

The Emergency Release connection will also be used in this example. Select the “Emergency Release to Reach” button (red circle in next figure). The cursor will then allow the user to select the reach segment to which the reservoir discharges will be routed (red arrow).



6.20

After the segment is selected, the following "Connection Properties" table will appear. Type in the values as they appear in the figure below.

6.21

Select the "Save" button to return to the "Reservoir Parameters" menu. Then select the "Done" button and exit from the reservoir menus following the same steps used in Section 6.2.

6.5 LAUNCH AND REVIEW RESERVOIR OUTPUTS

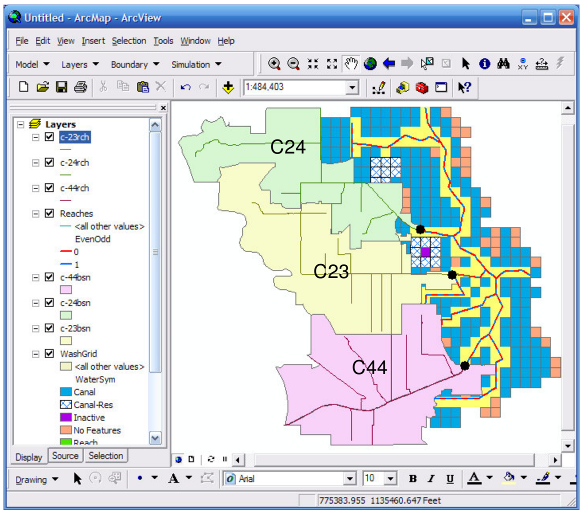
From the Simulation menu, select "run" to launch the simulation.

A DOS screen will appear and will continually update the simulation date as the simulation progresses. When the simulation is complete, press Enter on the keyboard and the DOS screen will disappear.

The remainder of this section is awaiting the final WaSh model outputs

Three upland basins of the St. Lucie Estuary Watershed drain to the St. Lucie Estuary. In this section, the measured flows from these basins are going to be added to the model reach network as point sources.

Figure 7.1 shows the SLE model with the upland basins and their associated reach networks added to the ArcGIS session. These are not incorporated into the model, but they illustrate the location that each upland basin reach will be attributed to the SLE model as a point source (black dots).



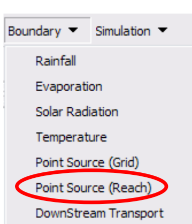
7.1

This view is not available part of the tutorial screen. It was generated using advanced GIS operations and is only provided to show the user where the point source points will be located.

The daily measured flows from each of these basins have been provided as input files as part of the tutorial files. The files are C24_discharge.txt, C23_discharge.txt, and C44_discharge.txt and are located in the SLE/configuration_files folder. The files contain both hydrodynamic and water quality point source daily inputs. For the simulation executed in this section, the water quality data is not necessary. However, the water quality data can be present in the file and will be ignored by the model unless a water quality simulation is added (adding a water quality simulation is discussed in Section 8.0).

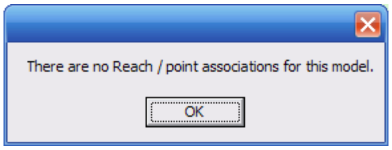
7.1 ADDING A POINT SOURCE

C24 basin discharge will be added first. From the Boundary menu, select the Point Source/Reach menu.



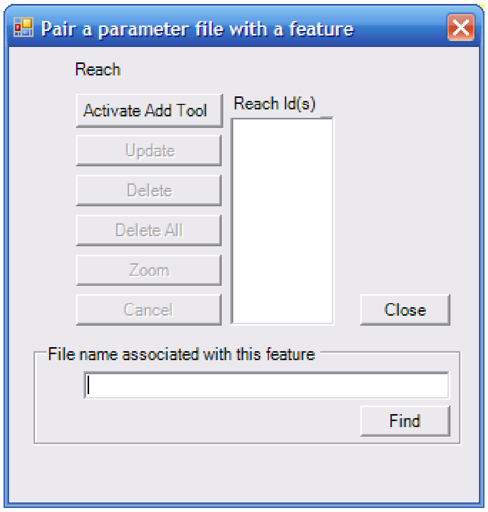
7.2

Since no point sources have been added, the following screen will appear. Select the “OK” button to continue.



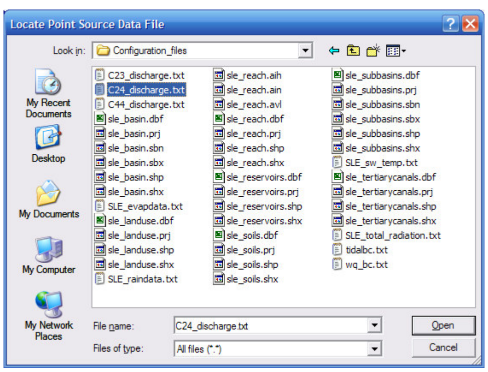
7.3

The following Reach Point Source screen will appear.



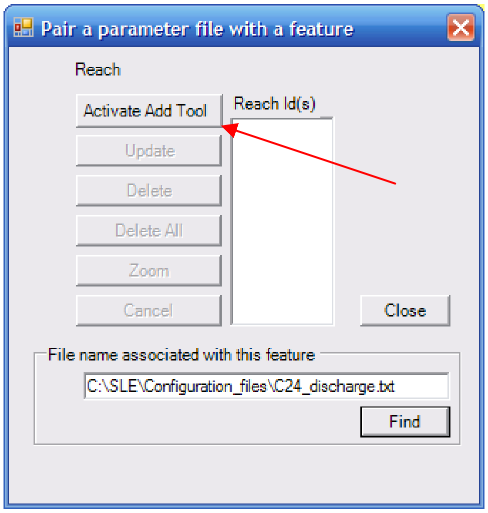
7.4

Use the "Find" button to open a browser and navigate to the SLE/configuration_files folder.



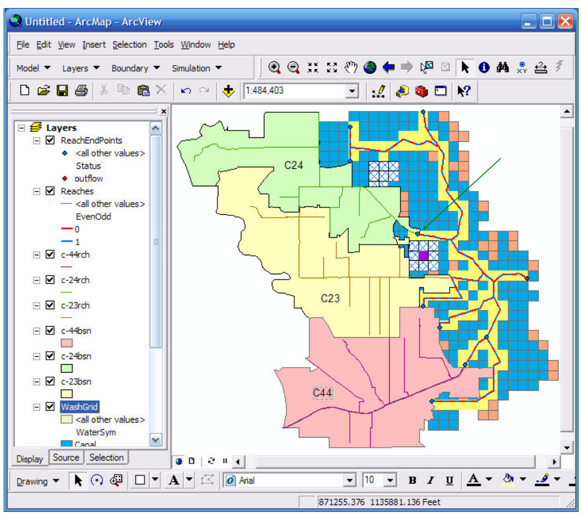
7.5

Select the C24_discharge.txt file from the menu and select "Open". The Point Source screen will reappear. Select the "Activate Add Tool" button.



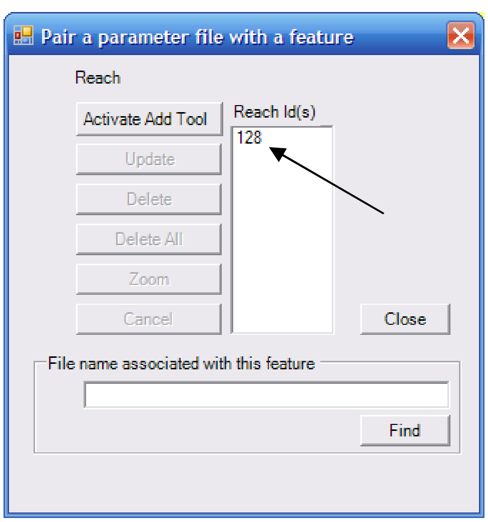
7.6

In the ArcGIS session, use the cursor tool to select the C24 basin outfall location indicated by arrow in the following figure.



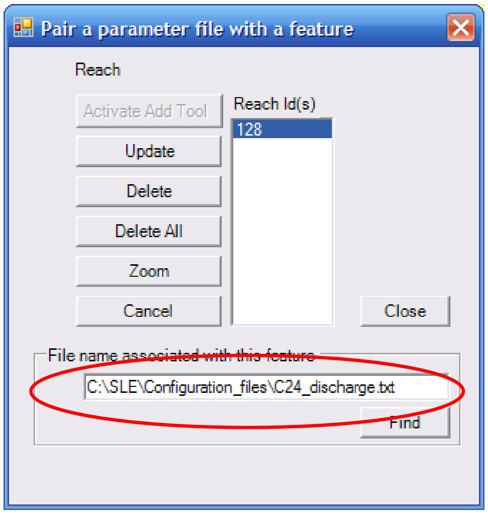
7.7

The file name will disappear from the box but a number will appear in the Reach ID box.



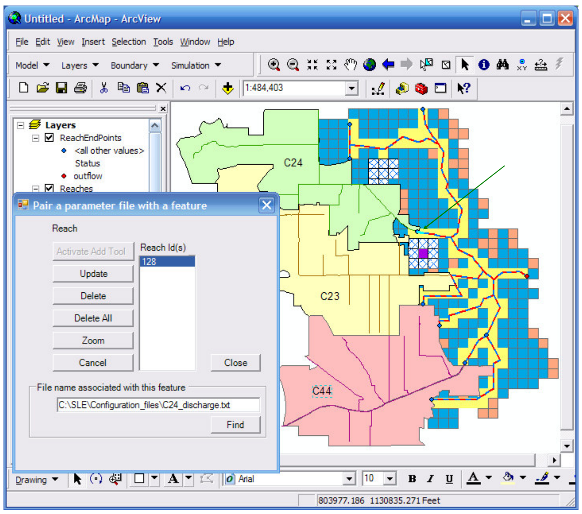
7.8

Highlight the Reach ID and the associated point source file will appear in the filename field.



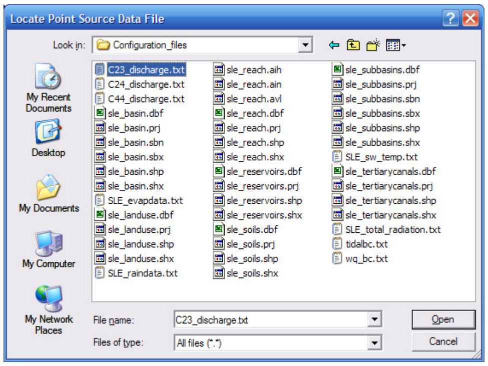
7.9

The proper reach segment will also be highlighted on the model grid (green arrow).



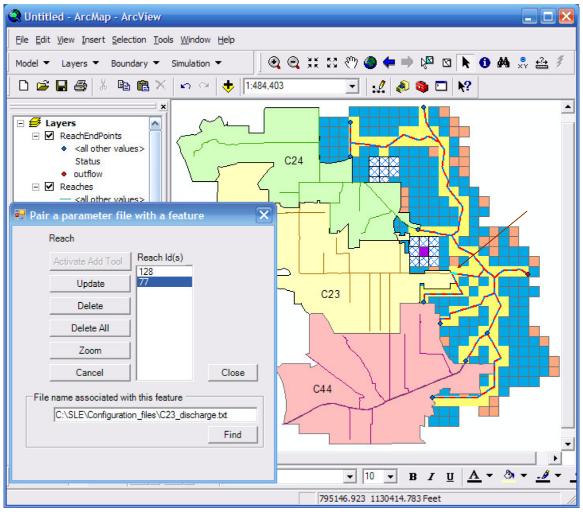
7.10

The same procedure will now be repeated to add the other two point sources. Select the "Cancel" button from the Point Source screen to remove the highlight from the C24 point source reach segment. Then select the "Find" button to open the file browser.



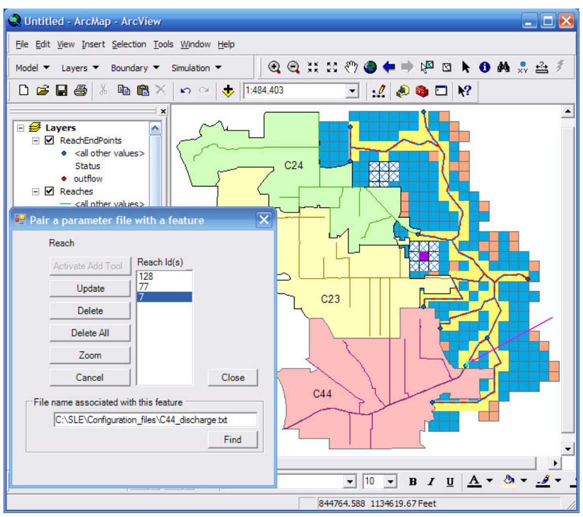
7.11

Find and select the C23_discharge.txt file and select the "Open" button. Select the "Activate Add Tool" and select the C23 Point source location as shown in the following figure.



7.12

Select the "Find" button, select the C44_discharge.txt file, select the "Activate Add Tool", select the C44 basin outfall reach segment, highlight the number "7" in the reach ID box and see the highlighted reach segment on the grid.



7.13

When all point sources have been added, select the "Update" button to save the point sources to the model.

Select the "Close" button to exit the Add Point Source menu.

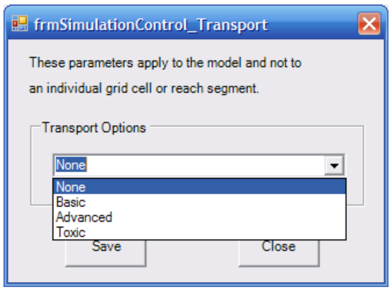
7.2 LAUNCH AND REVIEW THE OUTPUTS WITH POINT SOURCES ADDED

From the Simulation menu, select "run" to launch the simulation. A DOS screen will appear and will continually update the simulation date as the simulation progresses. When the simulation is complete, press Enter on the keyboard and the DOS screen will disappear.

The remainder of this section is awaiting the final WaSh model outputs

In this section, the advanced water quality simulation will be added to the model. The model will first be configured with a single vertical layer in the reach network and then with two layers. Also, some of the input parameters for the water quality simulation will be edited using the editing tools.

To initiate the advanced water quality simulation, select the “Transport” option in the Simulation menu. The following menu will appear, the SimulationControl_Transport menu. Open the drop-down menu and select the “Advanced” option.



8.1

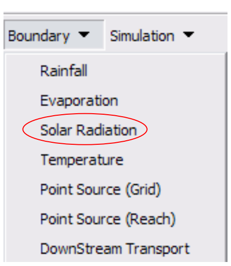
Select the “Save” button to exit the menu. The advanced water quality simulation will simulate concentrations of Ammonia (NH3), Nitrate (Nitr-N), inorganic phosphorus (IORG-P), phytoplankton carbon mass (Phyt-C), carbonaceous biologic oxygen demand (CBOD), dissolved oxygen (DissOx), and organic nitrogen (ON-N).

8.1 ADDITIONAL FORCING DATA INPUTS

The advanced water quality simulation requires additional forcing data to specify solar radiation and surface water temperature. The input files for these forcing data are available as part of this tutorial and are stored in the SLE/configuration_file folder.

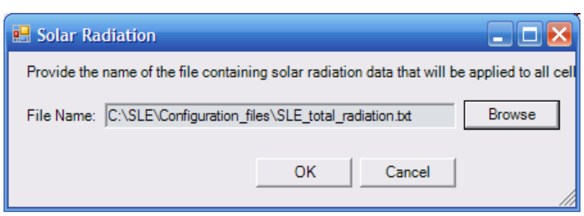
8.1.1 Total Radiation

Select the “Solar Radiation” option under the Boundary menu.



8.2

The following menu will appear:

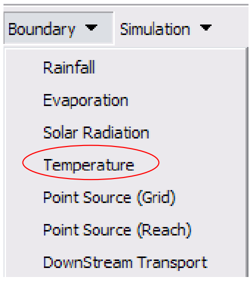


8.3

Use the browser to navigate to the SLE/configuration_files folder and select the SLE_total_radiation.txt file. Then select the "OK" button to exit the menu.

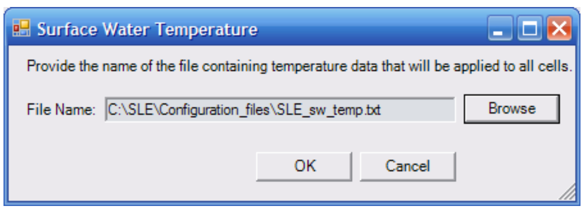
8.1.2 Surface Water Temperature

Select the "Temperature" option under the Boundary menu.



8.4

The following menu will appear:



8.5

Select the browser to navigate to the SLE/configuration_files folder and select the SLE_swl_temp.txt file. Then select the "OK" button to exit the menu.

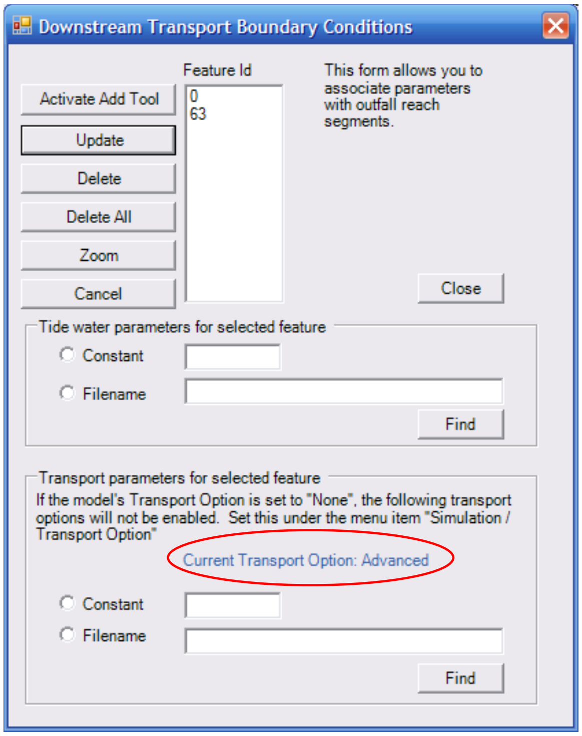
8.2 SINGLE-LAYER SIMULATION

The single layer reach network is the default when advanced water quality is initiated selected. Thus, there are no additional edits required to set the single layer option. However, the downstream boundary conditions needs to be set. Also in this section, the methods and tools available to edit model parameter values are demonstrated for some parameters.

8.2.1 Boundary Conditions

At the tidal boundary used in the hydrodynamic simulation, flow only has been entered into the reach network. It is therefore necessary to provide a downstream boundary condition for the advanced water quality parameters at that boundary, as well. An advanced water quality boundary conditions time series file has been provided as part of this tutorial and is located in the SLE/configuration_files folder. It contains a column for the year, month, day, and then eight columns for the eight advanced water quality constituents.

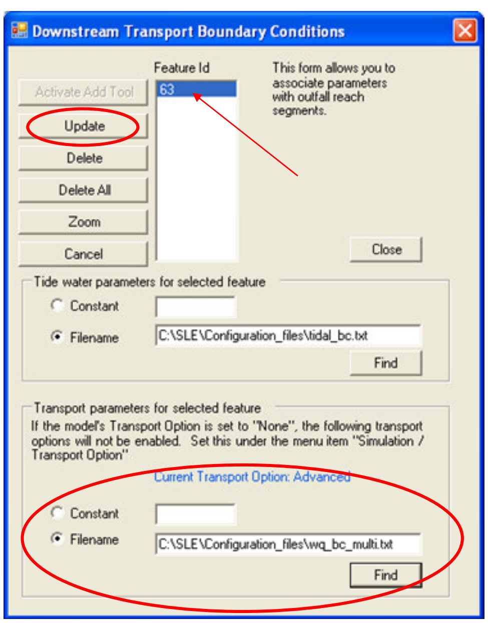
Select the Downstream Transport option from the Boundary menu. The Downstream Transport Boundary Option screen will appear.



8.6

The blue text indicates that the advanced Transport Option is selected and the boundary conditions fields are now active.

The reach segment for the tidal boundary condition has already been identified when the tidal boundary was configured in Section 3.6. To specify the advanced water quality boundary conditions, highlight the segment "63" in the Feature ID box. Then, select the "Find" button in the "Transport parameters for selected feature" section of the menu, which will launch a windows browser. Use the browser to navigate to the SLE/configurations_file folder, and select the wq_bc.txt file. Then select "OK" to exit the browser.



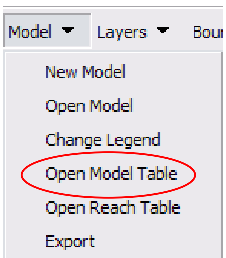
8.7

Select “Close” to exit the menu and save the advanced water quality boundary condition information.

8.2.2 Setting EMC Values

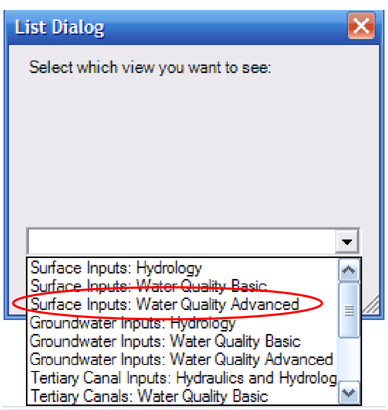
The event mean concentrations (EMCs) are values that set the constituent concentrations in the surface and groundwater discharges. They are often used as the key inputs in a water quality simulation and are based on local or regional measurements or values found in the literature.

The parameter value tables can be accessed through the “Open Model Table” in the Model menu. When the menus are opened for the first time, the WaSh model default values are loaded into the tables. This process may take a few minutes, but only occurs once for each table.



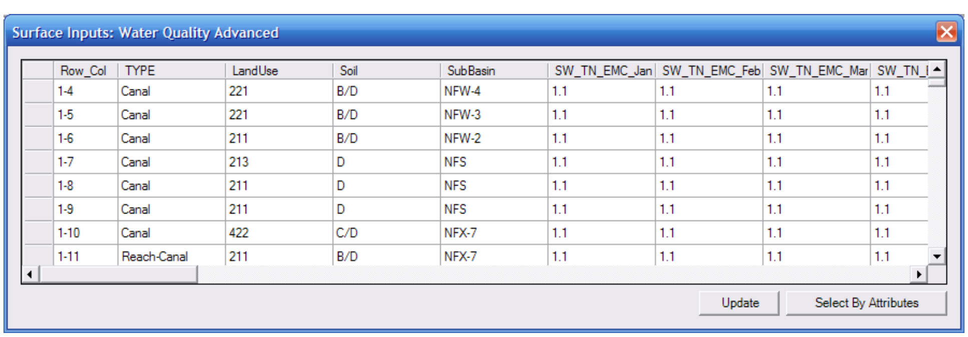
8.8

Select the “Open Model Table” option and the following screen will appear. Choose the “Surface Inputs: Water Quality Advanced” view, which is the table that contains the EMC values.



8.9

The cell editor window will appear. The cell editor displays basic cell information and the parameter values assigned to each cell for various model parameters.



Row_Col	TYPE	LandUse	Soil	SubBasin	SW_TN_EMC_Jan	SW_TN_EMC_Feb	SW_TN_EMC_Mar	SW_TN_I
1-4	Canal	221	B/D	NFW-4	1.1	1.1	1.1	1.1
1-5	Canal	221	B/D	NFW-3	1.1	1.1	1.1	1.1
1-6	Canal	211	B/D	NFW-2	1.1	1.1	1.1	1.1
1-7	Canal	213	D	NFS	1.1	1.1	1.1	1.1
1-8	Canal	211	D	NFS	1.1	1.1	1.1	1.1
1-9	Canal	211	D	NFS	1.1	1.1	1.1	1.1
1-10	Canal	422	C/D	NFX-7	1.1	1.1	1.1	1.1
1-11	Reach-Canal	211	B/D	NFX-7	1.1	1.1	1.1	1.1

8.10

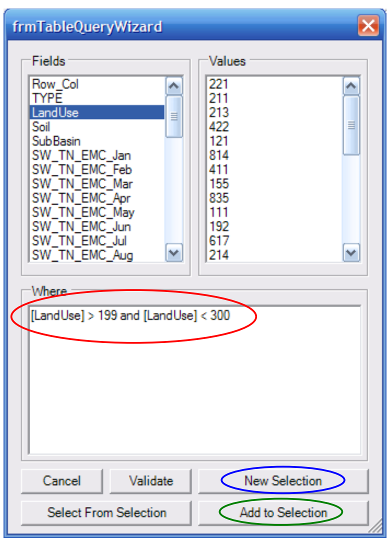
The role and function of each of these parameters are discussed in detail in the User's Manual, Section 6.2.1.

There are two basic steps in making edits to the parameter values. The first is identifying the cell or cells for which the changes are to be made. The second step is to identify the parameter and change it values. In the following section, various approaches for accomplishing these steps are described.

8.2.2.1 Selecting Cells by Land Use

In this example, the Total Nitrogen EMC values will be changed from their default values by assigning new, uniform values for all cells with agricultural land use. The WaSh model assigns twelve EMC values for each constituent to represent seasonal variations in the values. Values will need to be changed for each of the monthly EMC value.

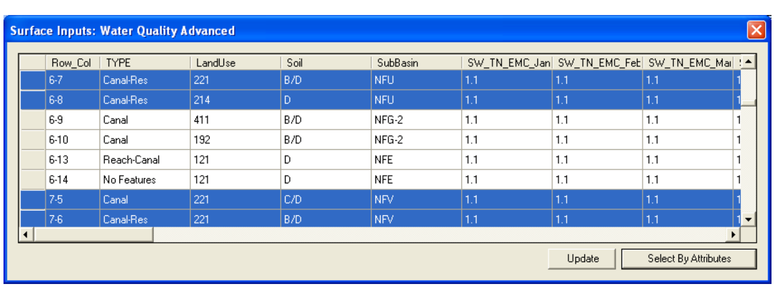
Select the "Select by Attributes" option in the bottom right corner of Figure 8.10 and the Table Query Wizard screen appears (Figure 8.11). A query will be constructed that will identify all of the agricultural cells, which are those with land use codes ranging from 200 to 299. Double click on the "LandUse" option in the Fields box. This will add "[landuse]" to the query window (i.e. "Where" window). Type in ">199 and" after "[landuse]". Then double-click on "LandUse" in the Fields box again. Type in "<300" after "[landuse]" (red circle). This query equation selects all landuses greater than 199 and less than 300.



8.11

Select the "Add to Selection" button (green circle) to make the cell selection.

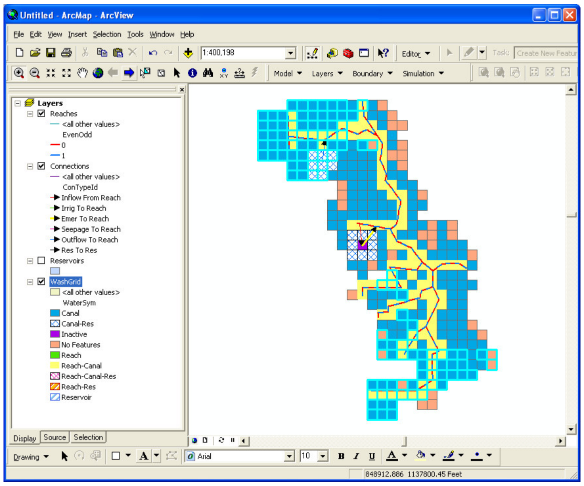
The selected cells will appear highlighted in blue in the editor window as shown in Figure 8.12.



Row_Col	TYPE	LandUse	Soil	SubBasin	SW_TN_EMC_Jan	SW_TN_EMC_Feb	SW_TN_EMC_Mar
6-7	Canal-Res	221	B/D	NFU	1.1	1.1	1.1
6-8	Canal-Res	214	D	NFU	1.1	1.1	1.1
6-9	Canal	411	B/D	NFG-2	1.1	1.1	1.1
6-10	Canal	192	B/D	NFG-2	1.1	1.1	1.1
6-13	Reach-Canal	121	D	NFE	1.1	1.1	1.1
6-14	No Features	121	D	NFE	1.1	1.1	1.1
7-5	Canal	221	C/D	NFV	1.1	1.1	1.1
7-6	Canal-Res	221	B/D	NFV	1.1	1.1	1.1

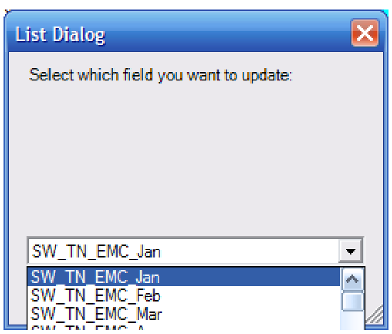
8.12

The selected cells are also highlighted with a cyan outline in the model grid display as shown in Figure 8.13.



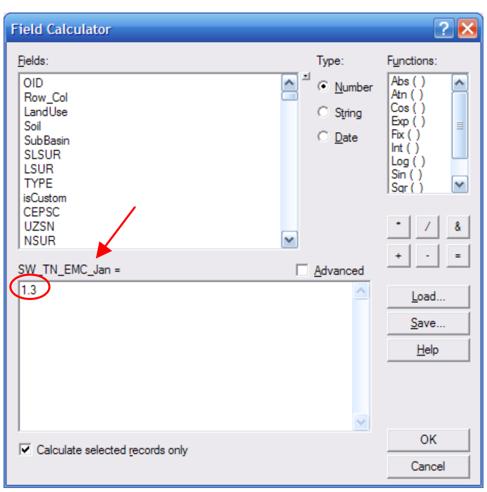
8.13

With the cells selected, the parameter values for those cells can now be edited. Select the "Update" button in the Surface Inputs Table (Figure 8.12) and the following menu appears. Highlight the SW_TN_EMC_Jan value (surface water total nitrogen EMC value for January) and selecting the "OK" button.



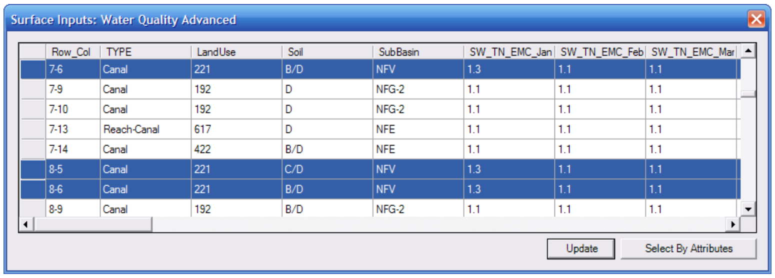
8.14

The ArcGIS Field Calculator will appear. The red arrow points to the parameter that is being edited. The box below the parameter should hold the new value for the parameter. Type 1.3 in the bottom box and select the "OK" button.



8.15

EMC values for all selected cells will now have a value of 1.3 for the SW_TN_EMC_Jan parameter.



8.16

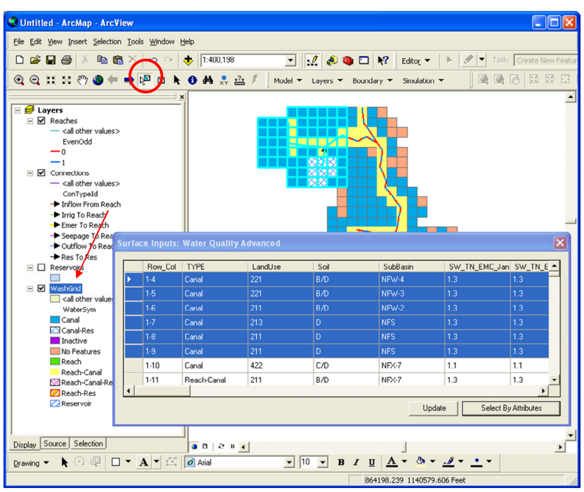
Repeat this process for the other 11 SW_TN_AMC values (Feb – Dec), setting the value to 1.3 for each parameter.

Note that if no cells were selected, all cells in the specified field will be updated with the indicated value. This method can be used effectively when uniform parameters are needed.

8.2.2.2 Selecting Cells by Location

In this example, the cells will be selected based on their geographic location, and the parameter values will be scaled down uniformly. This method is particularly useful when projecting the effects of using nutrient reductions in a system. For our example, a 50% nutrient reduction of Total Phosphorus will be simulated for a section of the North Fork area.

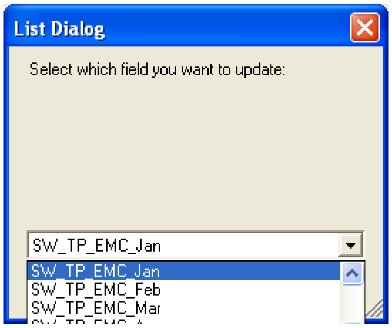
Instead of using the Select by Attributes option, grid cells will be selected directly from the ArcGIS session using the "Select Features" tool. With the WaSh Grid layer active (highlighted in blue at red arrow), pick the Select Features tool (red circle). With the cursor tool, grid cells may be selected by clicking on individual cells or using a polygon. For multiple selections, hold the shift key down. In the figure below, all of the cells above the Gordy Road structure have been chosen in the screen below, as indicated by the cyan outline around the cell edges. Use the select tool to select the same cells.



8.17

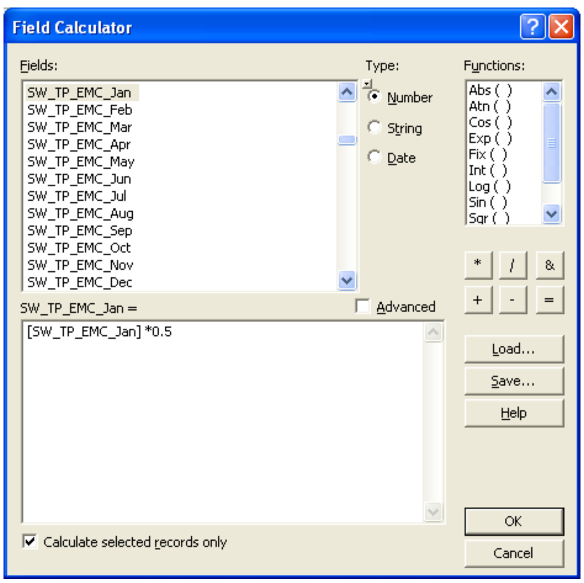
The selected cells will also be highlighted blue in the parameters table.

Select the "Update" button and the following screen will appear. Select the SW_TP_EMC_Jan (surface water total phosphorus EMC for January) and select "OK". This process has chosen the parameter SW_TP_EMC_Jan to be edited for all of the selected cells.



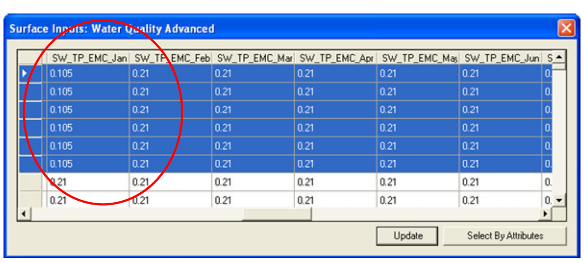
8.18

The field calculator will appear. Use the scroll bar in the Fields box to locate the SW_TP_EMC_JAN parameter. Double-click on the parameter name and it will appear in the edit box. After the parameter name, type *0.5 as shown in the figure below. Then select "OK".



8.19

Look in the Surface Input Table for Advanced Transport and see that January TP EMC has been reduced by 50%.



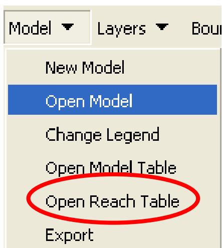
8.20

Repeat the above procedure for the other 11 parameter values (Feb through Dec), but do not exit the Table.

8.2.3 Setting Lateral Mixing

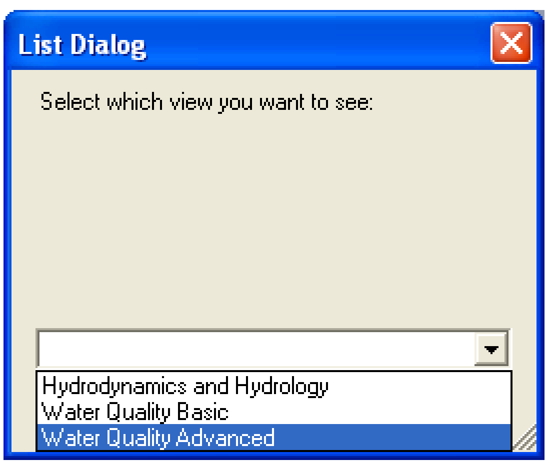
The default lateral mixing coefficient is modified in this section to reflect the differences in mixing in the rivers and the estuary. The default lateral mixing coefficient is1 ft^2/s, as shown in the figure below (Figure 8.23). In this example, the lateral mixing will be set to 5 ft^2/s for all reach segments, except for the estuary segments, which will be set to 100 ft^2/s. The changes will made by first changing all segments to 5 ft^2/s, and then only changing the estuary segments to 100 ft^2/s.

The reach parameter value tables can be accessed through the "Open Reach Table" in the Model menu. When the menus are opened for the first time, the WaSh model default values are loaded into the tables. This process may take a few minutes, but only occurs once for each table.



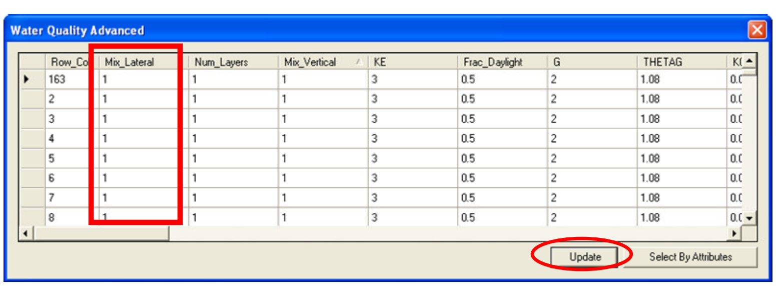
8.21

The following screen will allow the choice of transport type. Select the "Water Quality Advanced" option.



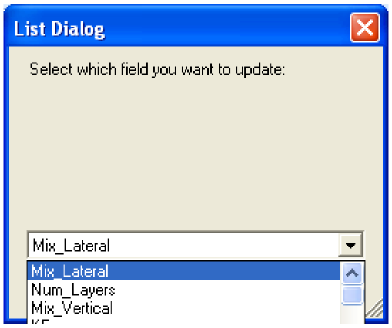
8.22

When the Open Reach Table is selected, the following table will appear. Note that the Mix_lateral parameter is set to 1 for all segments.



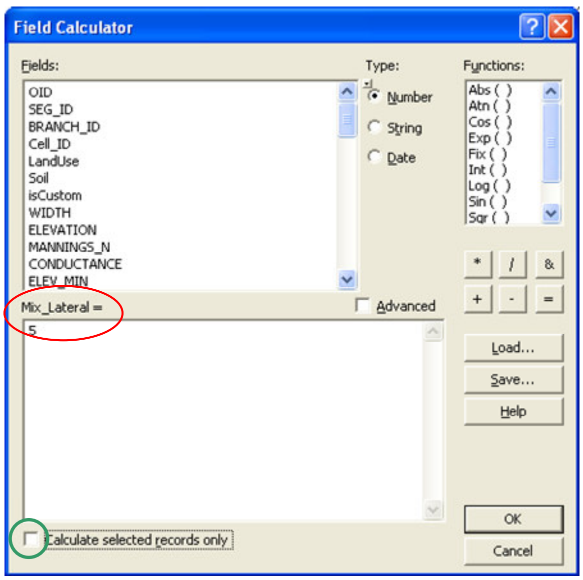
8.23

Select the "Update" button and the following screen will appear:



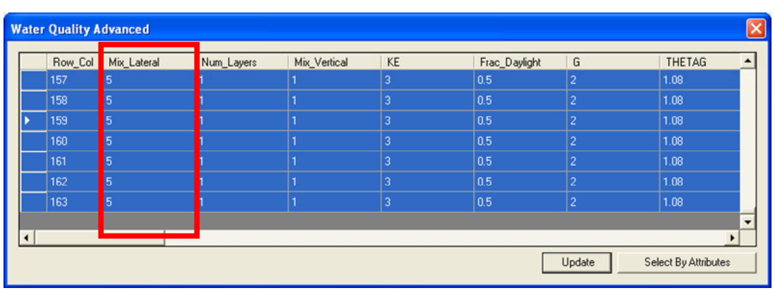
8.24

Select the "Mix_Lateral" parameter from the drop-down menu to open the Field Calculator. The Field Calculator identifies the "Mix_Lateral" parameter as the one to be edited. Type "5" in the equation box, as shown below. De-select the "Calculate selected records only" as shown in the green circle in the figure below. This will cause the edit to apply to all segments in the reach network. Select "OK".



8.25

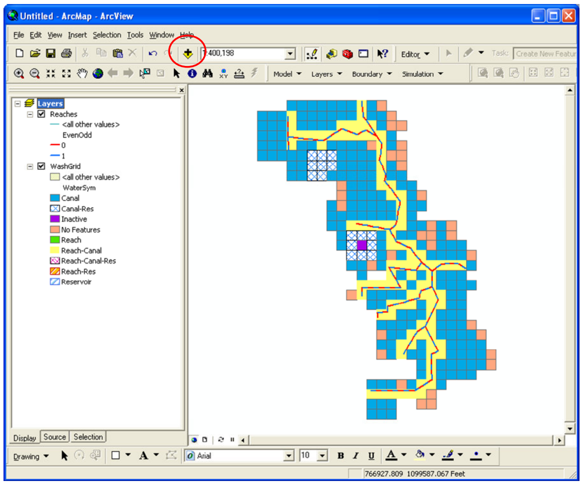
The Reach Table will appear as shown below, with all cells highlighted in blue and the values for Lateral Mixing changed to 5.



8.26

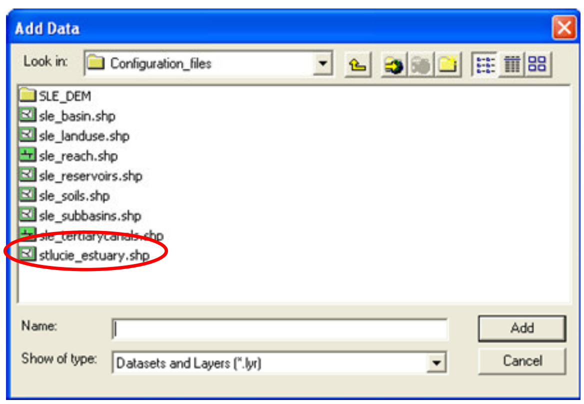
Next, the reach segments in the estuary will be changed to 100. In order to identify the segments that correspond to the estuary, another shapefile will be loaded and overlain on the model grid. The reach segments will then be selected using the ArcGIS "Select Feature" cursor using the shapefile that delineates the extents of the estuary as a guide.

First, choose the “Clear Selected Features” option from the Selection menu in the ArcGIS toolbar. This will remove the highlighting from the entire reach. Next, select the Add Layer button (red circle in figure below) to add the Estuary shapefile, which is located in the SLE/Configuration_files folder.



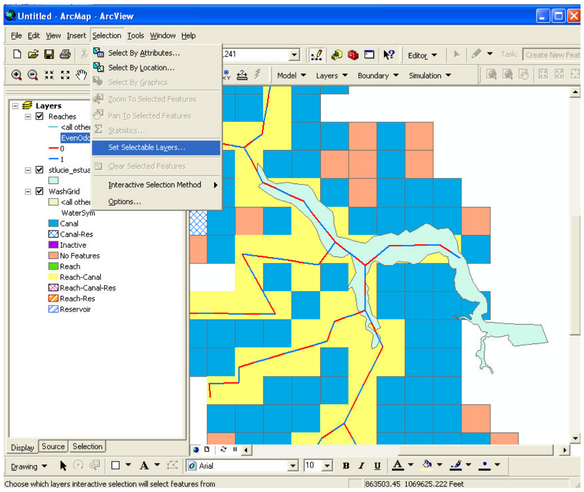
8.27

Using the browser, navigate to the SLE/configuration_files folder and select the stlucie_estuary.shp file as shown below. Select the “Add” button.



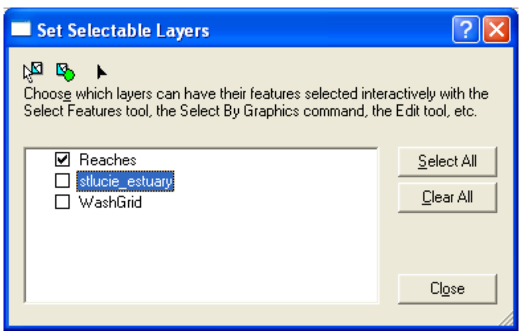
8.28

Use the Zoom tool to change the view to the estuary for easier segment selection. It is important to set the selectable features in ArcGIS at this point so that only the reach segments will be selected in the next step. Choose the “Select Selectable Layers” option from the Selection menu in the ArcGIS toolbar as shown below:



8.29

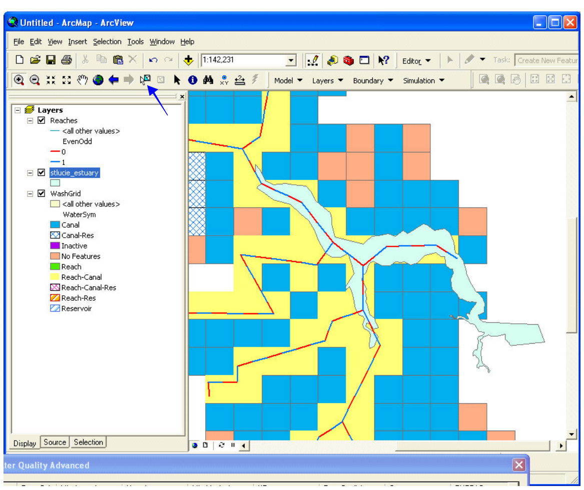
The following screen will appear. Check and un-check the layers so that only the "Reaches" layer is select as shown below:



8.30

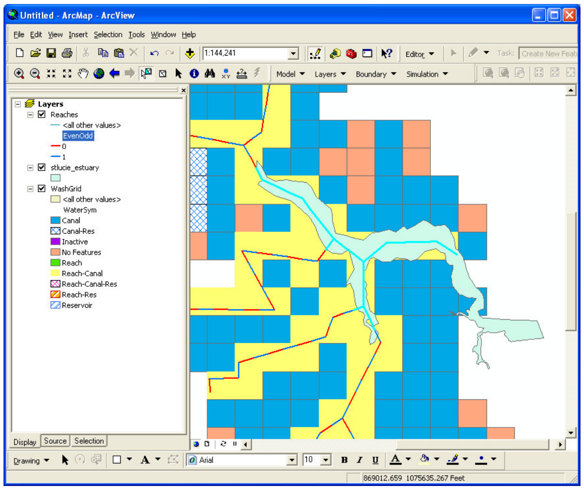
Select the "Close" button and return to the Arc session.

Next, select the "Select Features" tool. The tool is identified in the next figure by the blue arrow.



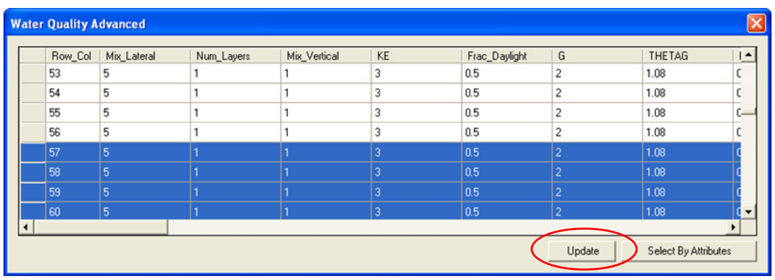
8.31

While holding down the "Shift" key, select each reach segment that resides within the boundary of the estuary polygon. When all segments are selected, the screen should appear as below. The segments will be highlighted in cyan.



8.32

Return to the Reach Water Quality Table and note that only the selected segments have been selected.

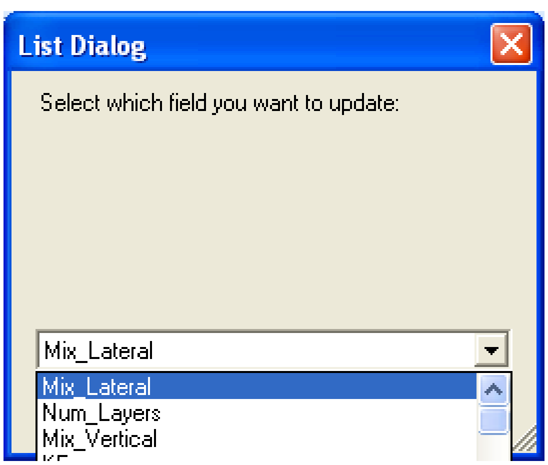

Water Quality Advanced

Row_Col	Mix_Lateral	Num_Layers	Mix_Vertical	KE	Frac_Daylight	G	THETAG
53	5	1	1	3	0.5	2	1.08
54	5	1	1	3	0.5	2	1.08
55	5	1	1	3	0.5	2	1.08
56	5	1	1	3	0.5	2	1.08
57	5	1	1	3	0.5	2	1.08
58	5	1	1	3	0.5	2	1.08
59	5	1	1	3	0.5	2	1.08
60	5	1	1	3	0.5	2	1.08

Update Select By Attributes

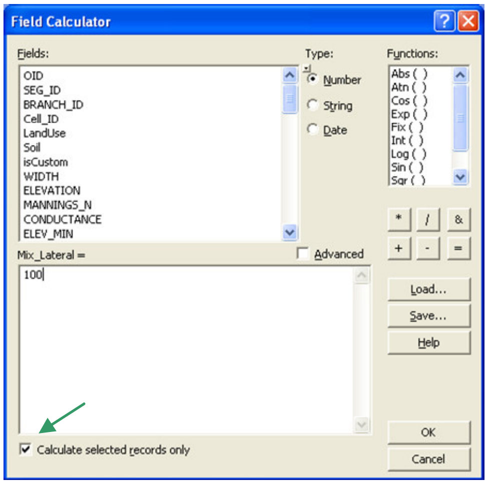
8.33

Select the “Update” button and then choose the Mix_lateral field from the pull-down menu as shown below:



8.34

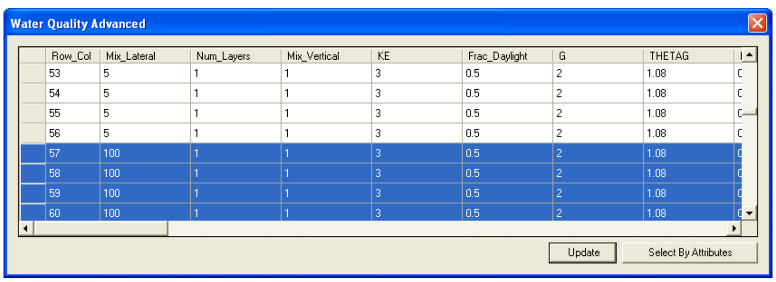
The Field calculator will appear allowing you to edit the selected records. Type “100” into the equation box. Check to make sure that the bottom option is checked (green arrow), indicating to the model that changes will be made to selected records only.



8.35

Select the "OK" button to exit the Field Calculator.

The Reach Table will now appear as below. All of the selected segments have a value of 100.



Row_Col	Mix_Lateral	Num_Layers	Mix_Vertical	KE	Frac_Daylight	G	THETAG
53	5	1	1	3	0.5	2	1.08
54	5	1	1	3	0.5	2	1.08
55	5	1	1	3	0.5	2	1.08
56	5	1	1	3	0.5	2	1.08
57	100	1	1	3	0.5	2	1.08
58	100	1	1	3	0.5	2	1.08
59	100	1	1	3	0.5	2	1.08
60	100	1	1	3	0.5	2	1.08

8.36

Select the close icon in the upper right hand corner to exit the table.

8.2.4 Launch and Review Outputs

From the Simulation menu, select "run" to launch the simulation. A DOS screen will appear and will continually update the simulation date as the simulation progresses. When the simulation is complete, press Enter on the keyboard and the DOS screen will disappear.

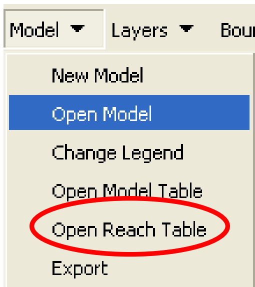
The remainder of this section is awaiting the final WaSh model outputs

8.3 TWO-LAYER SIMULATION

In this section the single-layer advanced water quality model will be converted to a two-layer model. The required changes are to include additional information at the tidal boundary and to specify the vertical mixing coefficient.

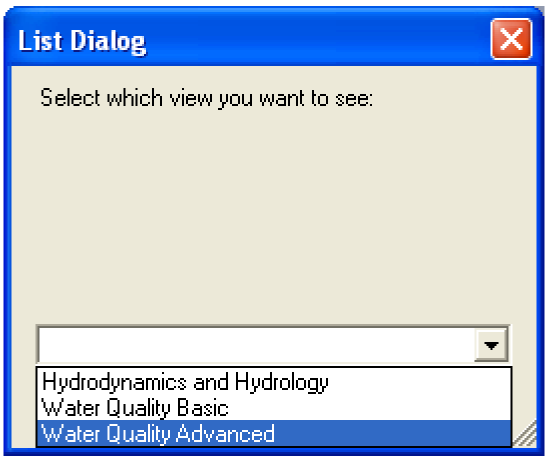
8.3.1 Specifying Two Layers

The two-layer option is specified by a parameter in the Reach Table. Select the "Open Reach Table" from the Model menu.



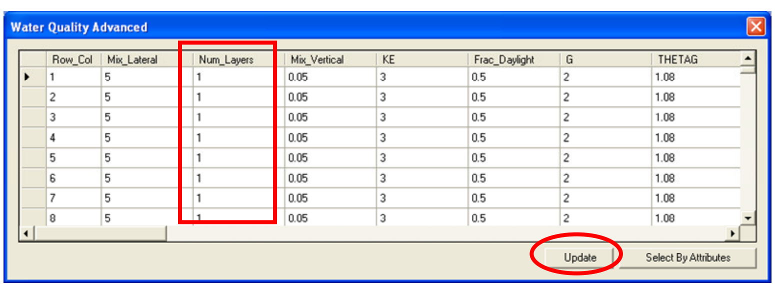
8.37

The following screen will allow the choice of transport type. Select the "Water Quality Advanced" option.



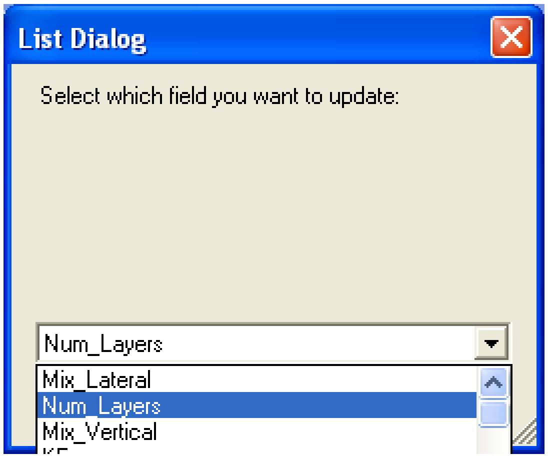
8.38

The "num_layers" field will be populated by the numeral 1 for each record, indicating that the model has been run previously for single layer simulations. All records will need to be changed to the numeral 2. Select the Update button.



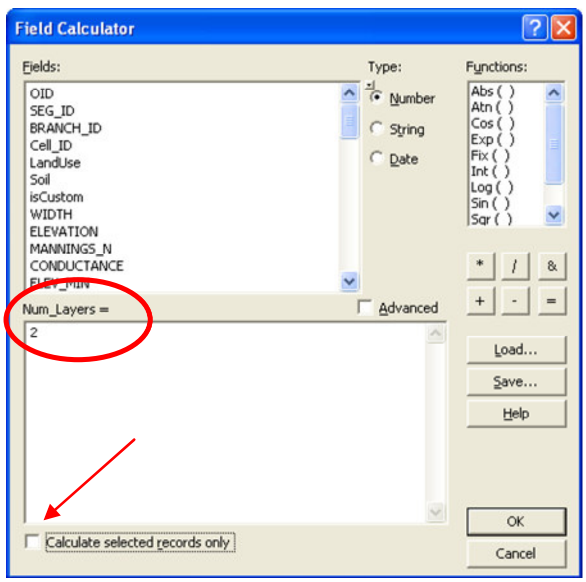
8.39

From the drop-down menu, choose the "Num_layer" field to edit.



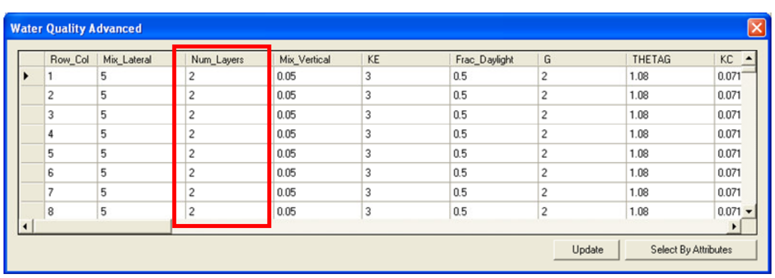
8.40

In the Field Calculator, type a 2 in the equation box. Make sure that the box in the bottom left corner is unchecked (red arrow), so that all layers will be edited rather than only selected records.



8.41

Select the "OK" button and the table will repopulate the Num_layers field with the numeral 2 in all records, indicating that the model will now be run as a multi-layer simulation.


Water Quality Advanced

Row_Col	Mix_Lateral	Num_Layers	Mix_Vertical	KE	Frac_Daylight	G	THETAG	KC
1	5	2	0.05	3	0.5	2	1.08	0.071
2	5	2	0.05	3	0.5	2	1.08	0.071
3	5	2	0.05	3	0.5	2	1.08	0.071
4	5	2	0.05	3	0.5	2	1.08	0.071
5	5	2	0.05	3	0.5	2	1.08	0.071
6	5	2	0.05	3	0.5	2	1.08	0.071
7	5	2	0.05	3	0.5	2	1.08	0.071
8	5	2	0.05	3	0.5	2	1.08	0.071

Update Select By Attributes

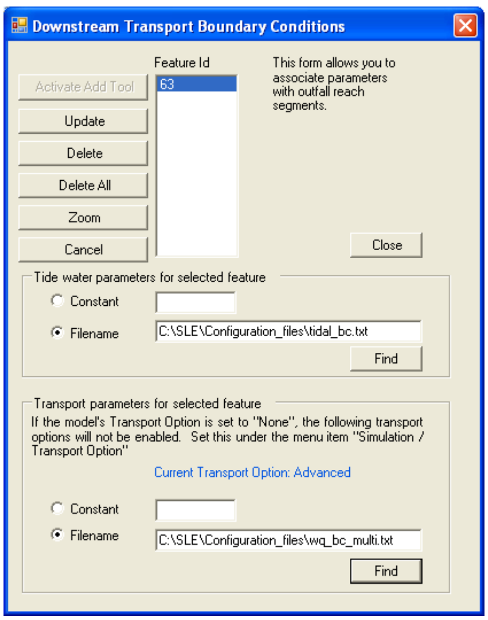
8.42

Exit out of the Reach Table using the exit icon in the upper right hand corner of the table.

8.3.2 Boundary Conditions

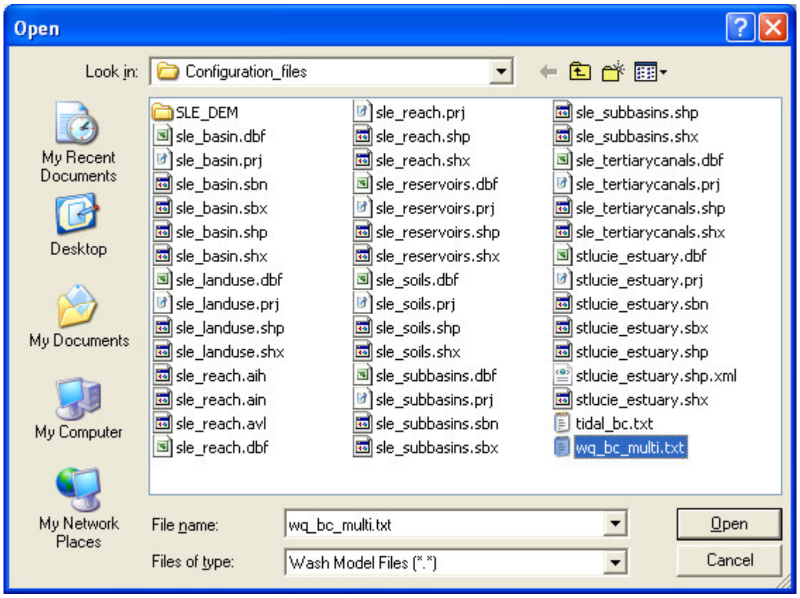
The advanced water quality boundary at the tidal boundary will need to be modified for the two-layer model application. The modification consists of using a different input time series file that has both upper and lower layer water quality values for the water quality constituents. An advanced water quality boundary condition time series file has been provided as part of this tutorial and is located in the SLE/configuration_files folder. It contains a column for the year, month, day, eight columns for the eight advanced water quality constituent values in the upper layer and then another eight columns for the eight advanced water quality constituent values in the lower layer.

Select the "Downstream Transport" option from the Boundary menu. The Downstream Transport Boundary Option screen will appear. To specify the advanced water quality boundary conditions, highlight the segment "63" in the Feature ID box.



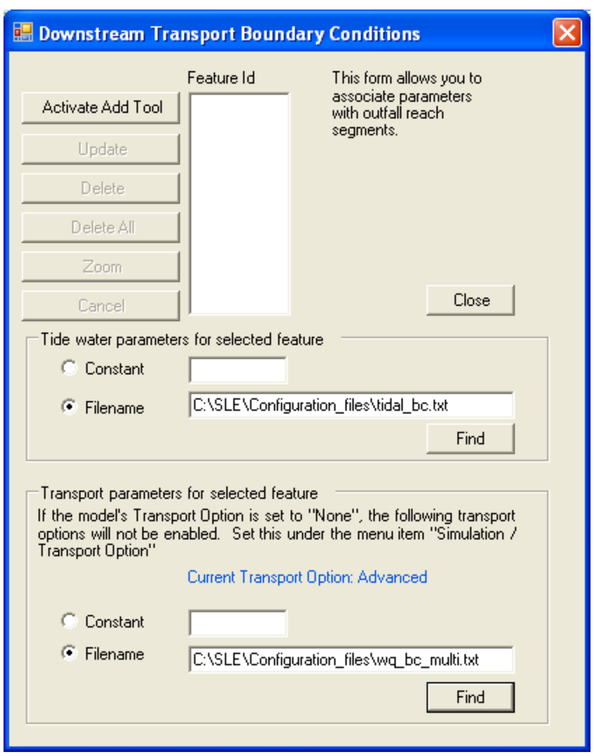
8.43

Then select the "Find" button in the "Transport parameters for selected feature" section of the menu, which will launch a windows browser.



8.44

Use the browser to navigate to the SLE/configurations_file folder, and select the wq_bc_multi.txt file. Then select the "Open" button to exit the browser. The Downstream Transport Boundary Conditions window will appear as follows:



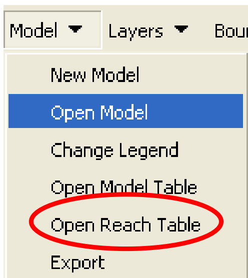
8.45

Select "Close" to exit the menu and save the advanced water quality boundary condition information.

8.3.3 Setting Vertical Mixing

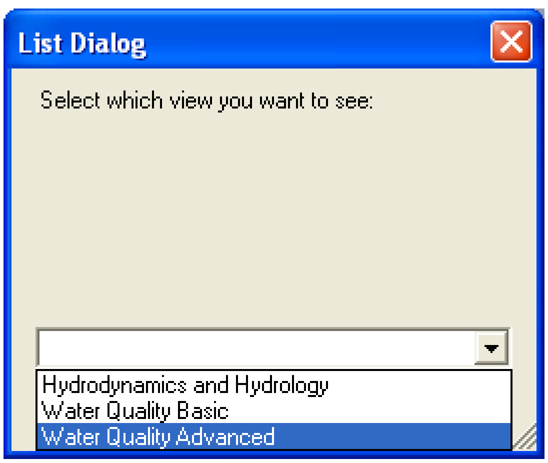
In the two-layer model applications of the advanced water quality simulations, the vertical mixing is a key parameter that controls the vertical transport of dissolved oxygen and other constituents. In this example, the vertical mixing is changed from its default value of 1 ft^2/s, to a value representative of the system, 0.05 ft^2/s.

Select the "Open Reach Table" from the Model menu.



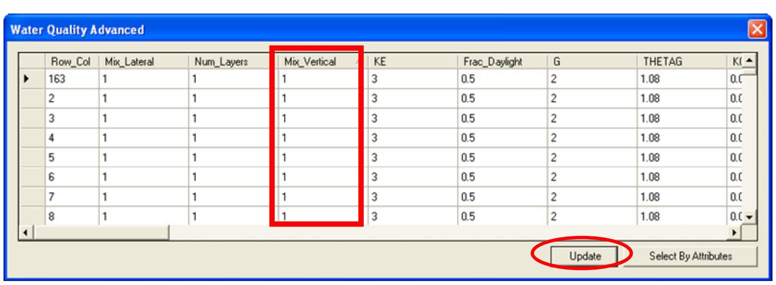
8.46

The following screen will allow the choice of transport type. Select the "Water Quality Advanced" option.



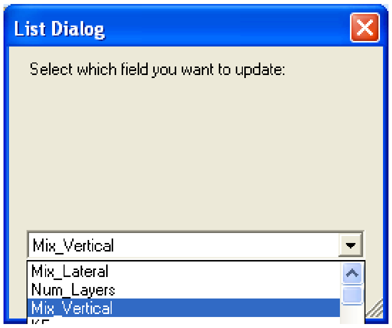
8.47

The "Mix_Vertical" parameter values are "1" for all segments, indicating that the model has been run previously for single-layer simulations. All records will need to be changed to "2".



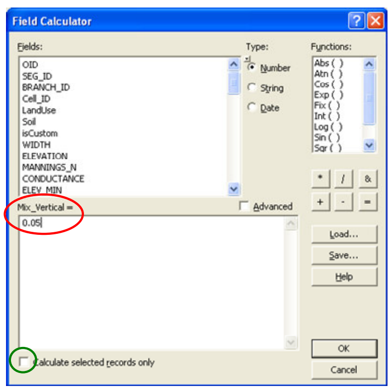
8.48

Select the "Update" button to open the following screen.



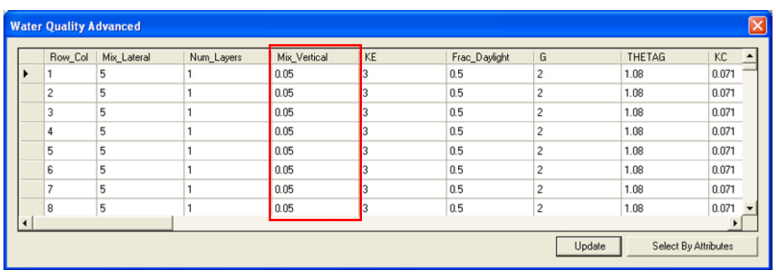
8.49

Select the Mix_Vertical parameter from the drop-down menu to open the Field Calculator. The red circle in the following figure identifies the Mix_Vertical parameter as the value to be edited. Type "0.05" into the equation box as shown in the figure below. Make sure to de-select the "Calculate selected records only" option at the bottom left corner of the screen (green circle) and ensure that the box is not checked. This will cause the change to be made to all segments, which is the intended result for this edit. Select "OK".



8.50

The Reach Table will appear as below, with all segment values for Mix_vertical changed to 0.05.



8.51

Exit the table by selecting the exit icon in the upper right hand corner.

8.3.4 Launch and Review Outputs

From the Simulation menu, select "run" to launch the simulation. A DOS screen will appear and will continually update the simulation date as the simulation progresses. When the simulation is complete, press Enter on the keyboard and the DOS screen will disappear.

The remainder of this section is awaiting the final WaSh model outputs

8.4 EXAMPLE BMP APPLICATIONS

The remainder of this section is awaiting the final WaSh model outputs

8.5 EXAMPLE TMDL APPLICATIONS (NUTRIENT REDUCTIONS)

The remainder of this section is awaiting the final WaSh model outputs

The remainder of this section is awaiting the final WaSh model outputs