United States
Environmental Protection Agency

Water Quality Analysis Simulation Program (WASP)

Graphical User Interface User's Guide

R E S E A R C H   A N D   D E V E L O P M E N T

EPA 600/R-??/???
December 2008
www.epa.gov

Graphical User Interface User's Guide

Supplement to Water Quality Analysis Simulation Program (WASP) User Documentation

Tim A. Wool
U.S. EPA, Region 4
Water Management Division
Atlanta, Georgia

Robert B. Ambrose, Jr., P.E.
U.S. EPA, Office of Research and Development
National Exposure Research Laboratory
Ecosystems Research Division
Athens, Georgia

Alex Comer
Tetra Tech
Atlanta, Georgia

U.S. Environmental Protection Agency
Office of Research and Development
Washington, DC 20460

NOTICE

The U.S. Environmental Protection Agency (EPA) through its Office of Research and Development (ORD) funded and managed the research described herein. It has been subjected to the Agency's peer and administrative review and has been approved for publication as an EPA document. Mention of trade names or commercial products does not constitute endorsement or recommendation for use.

Table of Contents

List of Figures

1. Introduction

The WASP graphical user interface (GUI) provides an environment for the user to develop input datasets to be used with the current version of the US Environmental Protection Agency's Water Quality Analysis Simulation Program (WASP). The GUI provides access to all of the water quality modules available with WASP.

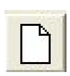
1.1. Installation

The WASP installation is accomplished much like any other Windows software installation. To initiate the installation:

1. Download the latest version of WASP from: www.epa.gov/athens/wwqtsc
2. Execute the SETUP program
3. Follow the prompts
4. Follow the instructions on the screen prompts to complete the installation.

1.2. Tool Bar Definition

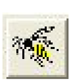
When the user first loads WASP a toolbar is displayed. This toolbar allows the user to navigate the different options and data entry forms of the program. Typically, the user builds input datasets accessing the toolbar icons from left to right. If a toolbar icon is visible but not colored, this indicates that the function is not yet available. This typically means that some prerequisite was not met yet.



This icon instructs the user interface to create a new file. Any data in the preprocessor will be lost.

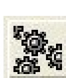
This icon allows the retrieval of a previously created model input file

This saves the active file to disk. Note that Save-as is available from the File Menu structure.

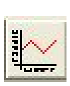


This executes the appropriate model based upon the currently loaded input file.

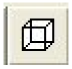
This icon is only available when the model is actually running. The user can abort the model simulation by pressing this icon. NOTE: It may take several minutes for the model to abort.

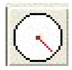


Model Parameterization (Page 9)

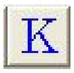


Model Simulation Result Interval (Page 16)

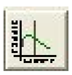
 **Model Segment Definition Screen (Page 17)**

 **Model System Definition (Page 14)**

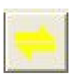
 **Segment Parameter Scale Factors (Page 22)**

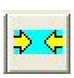
 **Model Kinetic Constant Definition (Page 32)**

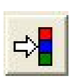
 **Waste Load Time Series Definition (Page 28)**

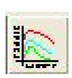
 **Environmental Time Series Definition (Page 30)**

 **Flow Data Entry (Page 25)**

 **Dispersion Data Entry (Page 23)**

 **Boundary Condition Time Series (Page 27)**

 **WASP Network Import Function (Page 47)**

 **This loads the graphical post processor. If the user has a project or model input file selected this information is passed to the post processor. (56)**

> The user has the option of using the Toolbar icons or the pull down Menus

1.3. File Menu

Because WASP has changed the methods in which model input data is stored the user may have to import old datasets in the new framework. Old WASP input files had an extension of INP, which stood for input file. These old style input files were ASCII formatted files that could be read by most word processors and utility text editors. WASP still stores the model-input data in individual files, but now they have the extension WIF, WASP input file. The new style input file is binary which allows for rapid saving/retrieving of information. The preprocessor can only view this file in a meaningful manner. WASP also supports a Project File format where the user can provide other WASP related files. Project files are edited from the project menu item.

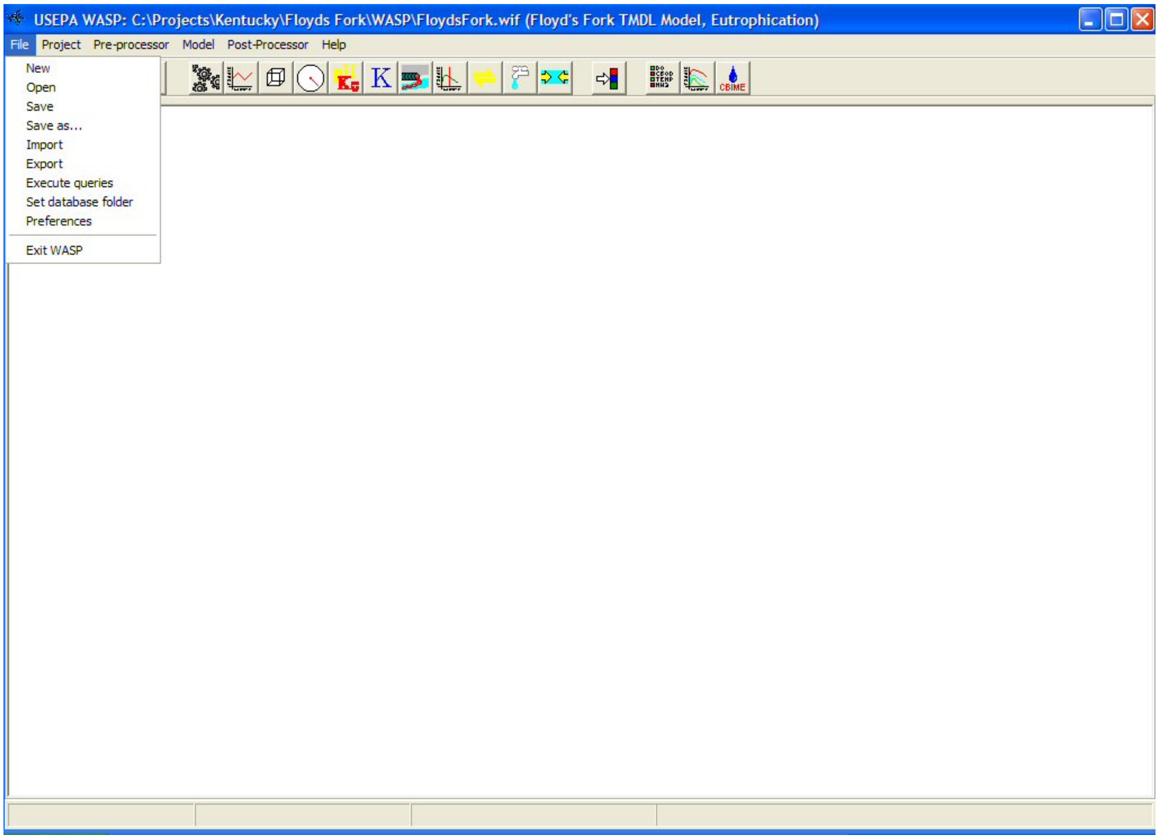


Figure 1-1 File Menu

1.3.1. New

This option initiates a new input file. All information currently loaded in the interface will be lost.

1.3.2. Open

This option opens a previously created input file

1.3.3. Save

This option saves the currently loaded input file.

1.3.4. Save as

This option allows the user to save the currently loaded input file to another filename.

1.3.5. Importing ASCII Files

This option imports an ASCII input file. There are two different file structures supported by the import function:

1. Single File Format – all data is contained in a single ASCII file. It is required that this filename have the extension *.INP. If you use the export function and select *.INP this file will be created.
2. Control File Format – data is imported from a set of files via control file. It is required that this filename have the extension *.WNF. These files have the same content and format as the WASP Network Import Function. If you use the export function and select *.INP this file will be created.

1.3.6. Exporting ASCII Files

This option exports an ASCII input file. There are two different file structures supported by the export function:

1. Single File Format – all data is contained in a single ASCII file. It is required that this filename have the extension *.INP.
2. Control File Format – data is imported from a set of files via control file. It is required that this filename have the extension *.WNF. These files have the same content and format as the WASP Network Import Function.

> User needs to have a very good understanding of WASP input requirements to use these functions

1.3.7. Execute Queries

This function is accessed from the file pull down menu (Figure 1-2). The purpose of this function is if the user developed their WASP input dataset using Live Queries. When the Live Query option is used the time series data is not actually stored in the WASP input file but remains in the external data source (database, spreadsheet). If the user wanted to send a WASP input file to someone the live queries would need to be converted to imported queries where the data is actually brought into the WASP input file.

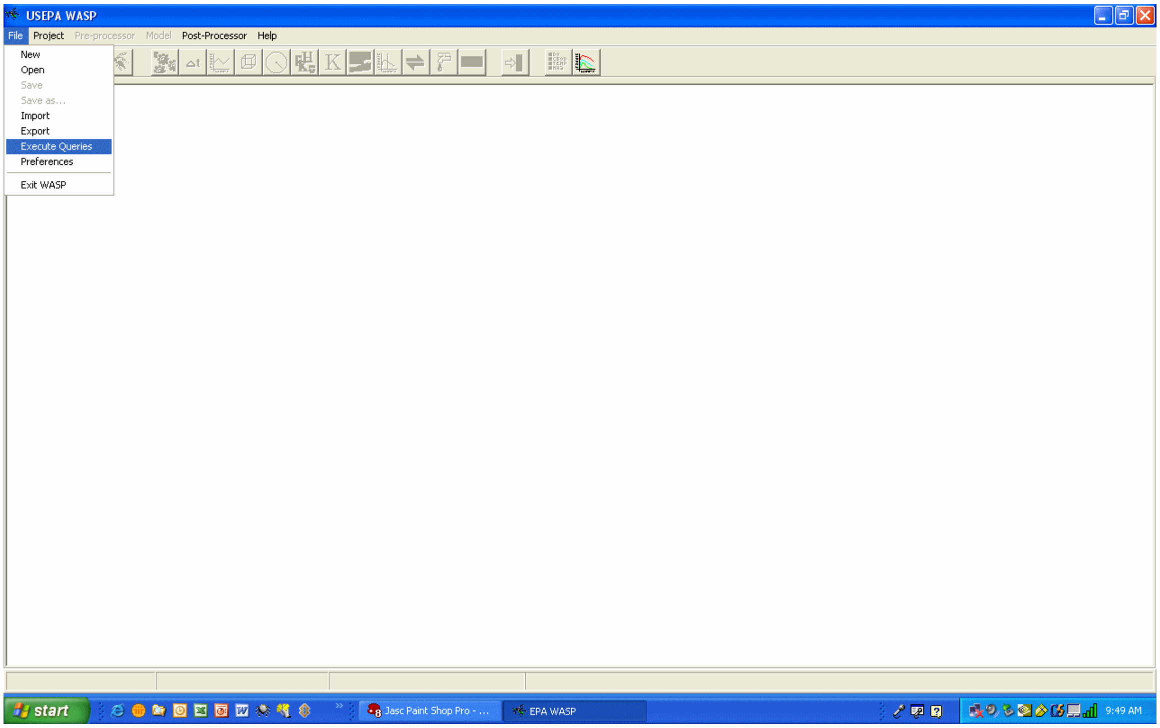


Figure 1-2 Time Series Conversion from Deferred to Imported

This option will convert all live queries; there is no provision to select the queries to be converted. The user would have to go to the individual time series screens and use the import option to do accomplish this task.

> Use this option to send a WASP Input File to someone else where the time series data are linked to external data sources

1.3.8. Database Folder

The WASP interface stores information for each of the model types in a series of database files (Figure 1-3). This option allows the database to be placed anywhere on your system. By default it is stored in a sub directory \database where WASP is installed.

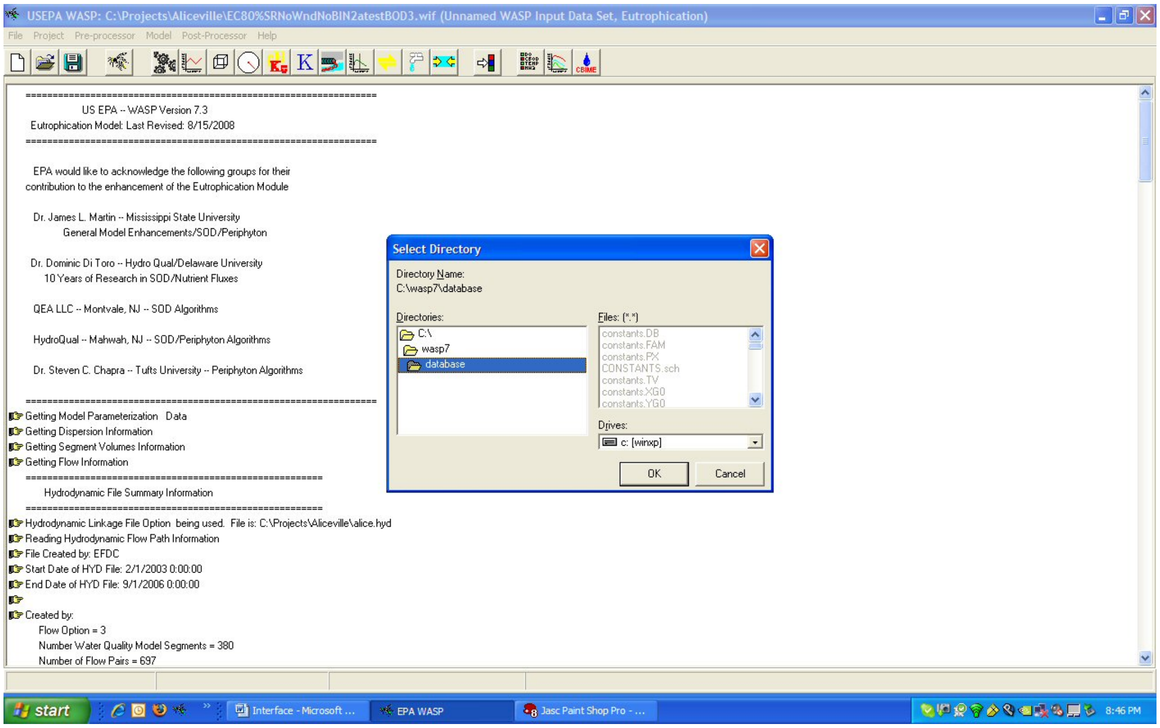


Figure 1-3 Database Folder Selection

1.3.9. User Preferences

The user has the ability to set several options within WASP (Figure 1-4). The first option is whether to display a condensed version of the toolbar or the complete toolbar. The user also has the ability to enable logging. This option is used for debugging purposes only. The logging function will generate a logging of all communications between the WASP program and the model DLL's.

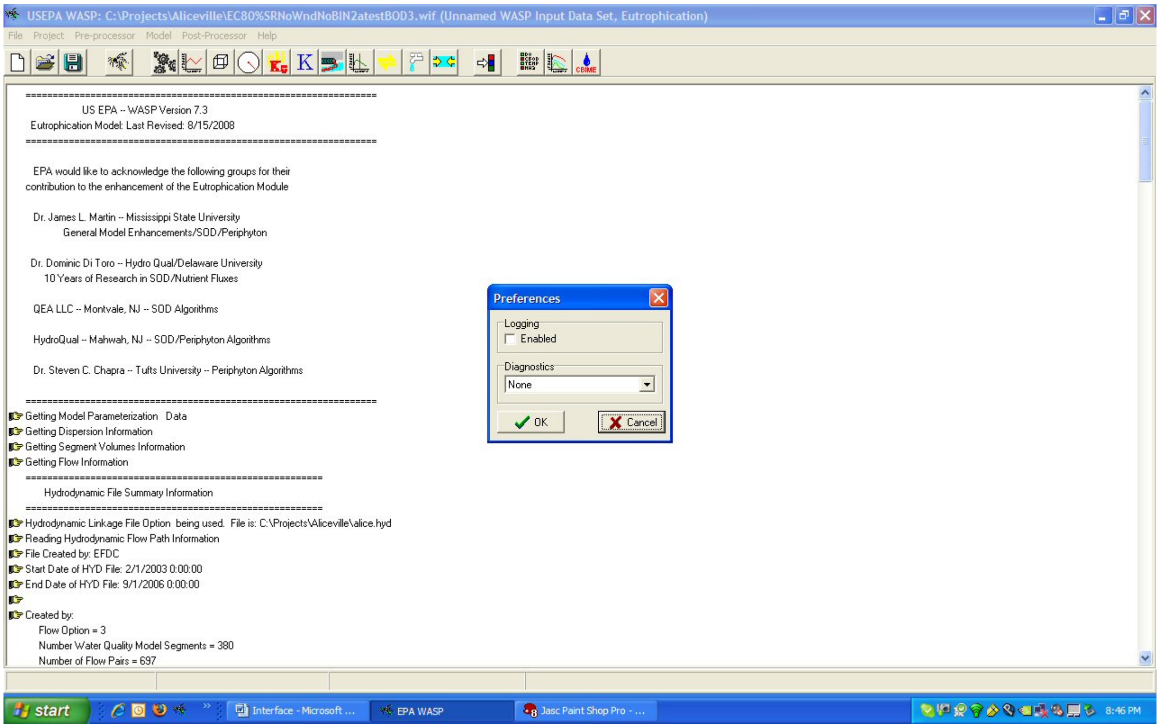


Figure 1-4 User Preferences

1.4. Project Files

The user can develop WASP input datasets without ever using the project file option. The Project file allows the user to specify in one place all of the files that are used for a given input/output file. The user can create a project file by selecting Edit Project from the Project Menu.

> Note: The Project Information is Stored in the Input File

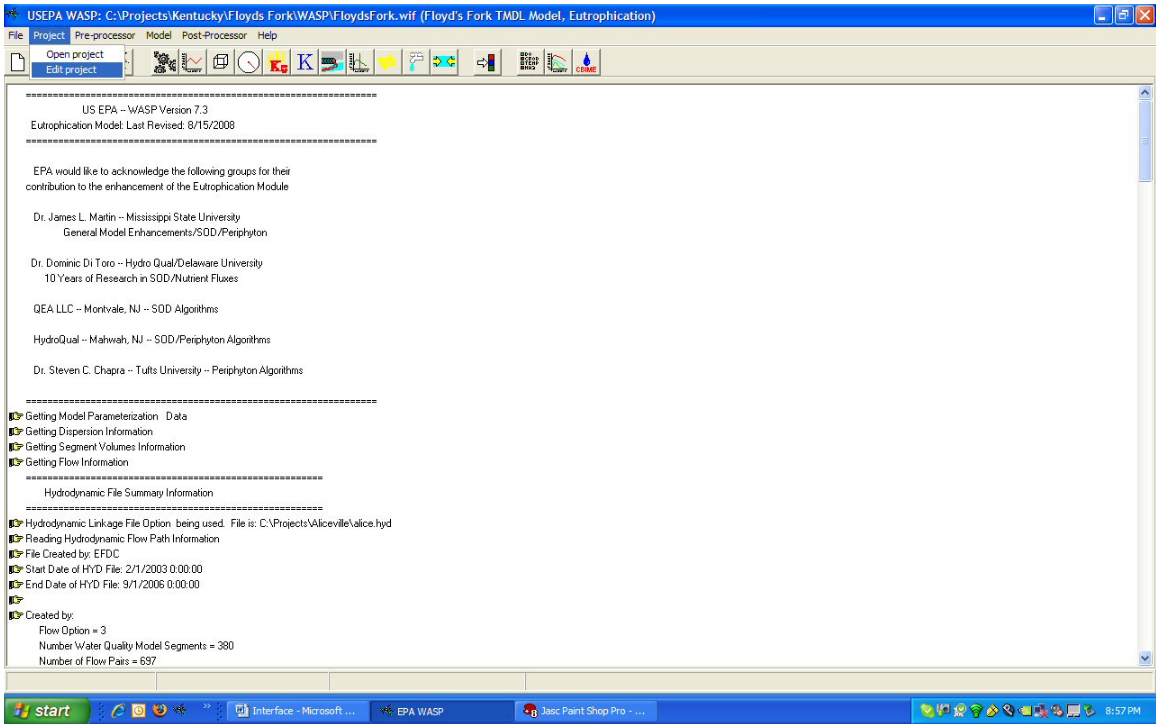


Figure 1-5 WASP Project Menu

There are five types of files that can be added to the project menu:

*.BMD – Any WASP/EFDC/EPD-RIV1 output file

*.DB – database files containing observed data

*.SHP – ArcInfo/ARCView shape files.

*.CLF – Curve Layout File created in MOVEM

*.SLF -- Spatial Plot Layout File created in MOVEM

When the post-processor is loaded the associated result file for the given WIF, any BMD, DB, SHP, CLF and SLF files are automatically read in.

1.4.1. Edit

The edit menu item allows the user to modify the contents of the opened project file. The users can remove/add files to the project.

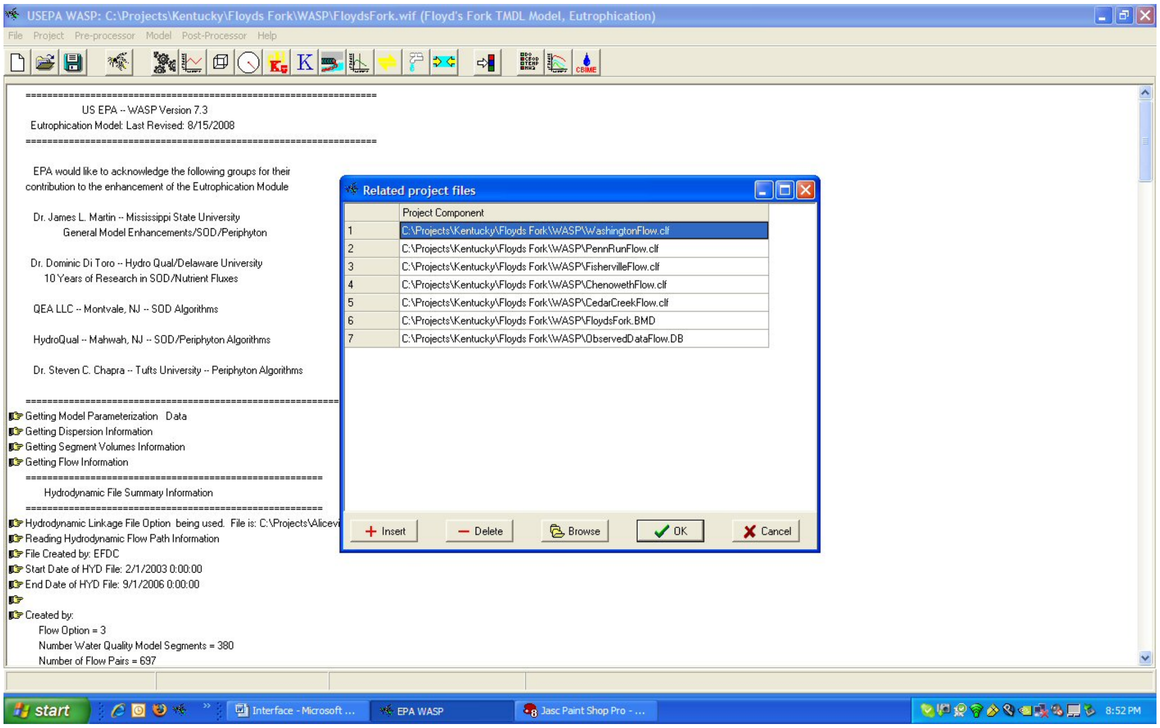


Figure 1-6 Project File Definition

1.5. Input Parameterization

When creating a new input dataset the input parameterization data entry form is the first one that needs to be completed. This form provides basic information that is needed by the program to parameterize the other data entry forms that follow. This screen informs the program what type of WASP file you are going to be creating.

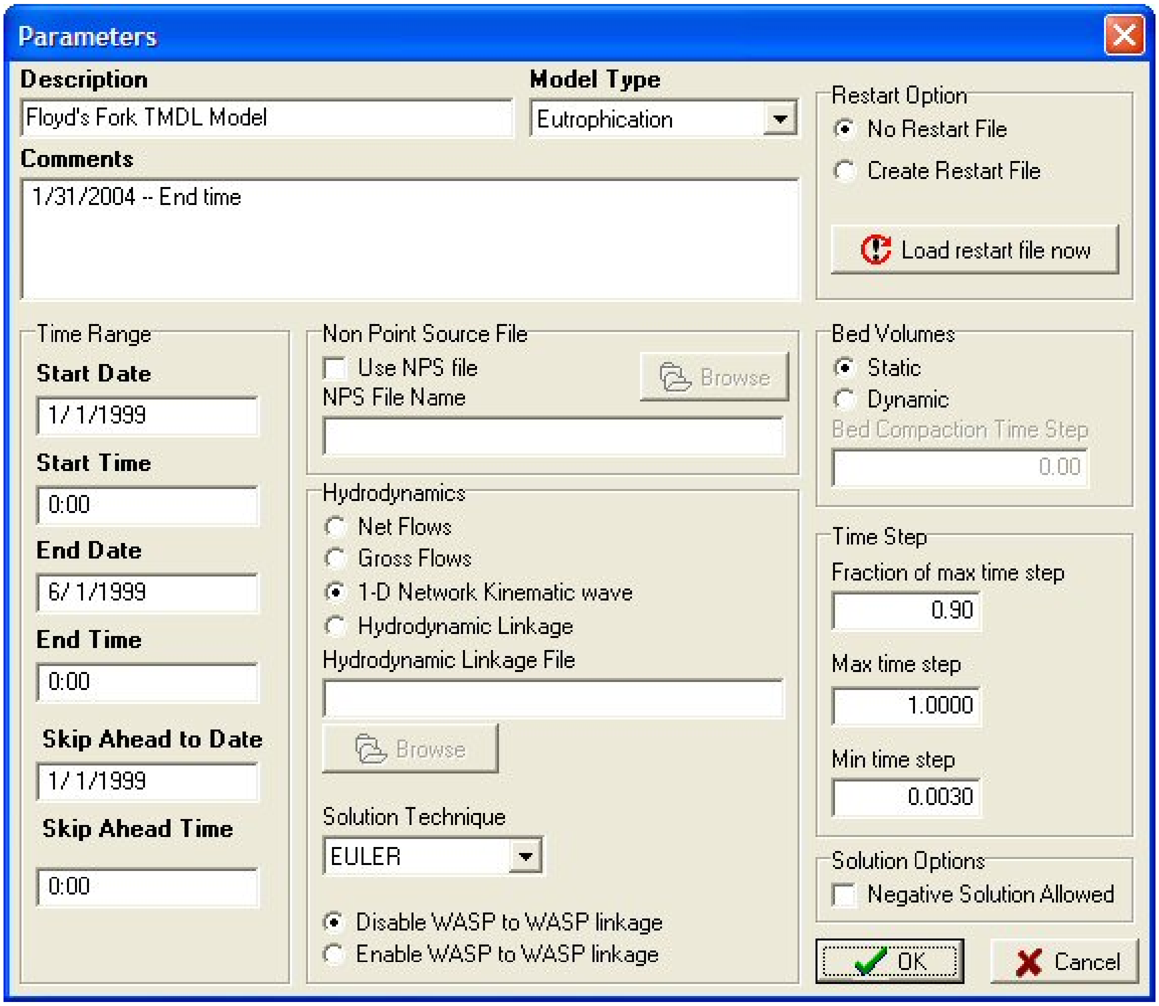


Figure 1-7 Dataset Parameterization

1.5.1. Data Set Description

This field provides a one-line descriptor for the defined input data file. This descriptor is displayed on the caption line of the main WASP window.

1.5.2. Model Type

The model type dialog box allows the user to specify which WASP model type the input dataset is being created. Setting the model type parameterizes WASP for that particular model type. Note that if you define a model as one type and change types later all model type specific data will be re-initialized (Time Functions, Kinetics, Parameters, Boundaries, Initial Conditions, Loads). Select the model type in the drop down picklist.

1.5.3. Comments

The dialog box provides space for the user to annotate important information about the dataset. The model does not need this information.

1.5.4. Restart Options

The methods used by WASP to read and create restart files have changed substantially in this version. In previous versions the user would have selected Create Restart File, for WASP to write the final conditions of the simulation to an output file. This is true for the current version as well. If the user wants to restart a simulation with the final conditions of previous simulation this radio must be set. At the end of the WASP simulation a restart file with the same name as the WIF except with the extension *.RST will be saved. With the current release of WASP if the user wants to use a restart file they simple click on the Load Restart File button, this will allow the user to browse to whatever restart file they want to use. Once the file is selected and the user clicks on the Okay button, the restart file is opened up and segment volumes and state variable initial conditions are reset to the values in the user selected *.RST file.

1.5.5. Date and Times

The previous versions of WASP did not require that the model time functions be represented in Gregorian date format. WASP requires all time functions be represented in Gregorian fashion (mm/dd/yr hh:mm:ss).

Start Time

The start time dialog is used to define the Gregorian date and time for the start of the simulation or time period being consider in the model input files. .user in the Start Time dialog box must specify the starting date and time. This date and time correspond to time zero in the old version of the model.

> Note the user can user the Skip Time function to move the start time further into the simulation

End Time

The end time dialog is used to define the Gregorian date and time when the simulation will end. End time use to be defined in the timestep definition screen that no longer exists.

Skip Time

This is a new addition to the WASP interface that allows the user to skip to any portion of the simulation and/or the selected loaded hydrodynamic linkage file. When the user selects a hydrodynamic linkage file the start time and end time of the file is read and the interface automatically sets the beginning and end time to these values. It is best that the user build all of the time series (environmental, boundary and dispersion) to cover this full range of time. Was the WIF is built the user can set a date and time to skip the simulation from the start time set in the hydrodynamic file. This is handy for using the whole hydrodynamic linkage file for calibration and verification, and then uses a small portion of the hydrodynamic linkage file for scenario analysis. It could be the critical time period that will be used for the waste load allocation or TMDL. The start time of the simulation should still be set the beginning time in the hydrodynamic linkage file. The user can change the end time of the simulation by changing the last date/time pair in the Time Step screen.

1.5.6. Non-Point Source File

The non-point source file is an external file that contains a time-series of loads (kg/day) for a given segment and system. This file is typically created either by the user manually or using other software like the Stormwater Management Model (SWMM) in conjunction with the Linked Watershed/Waterbody Model. This file can be used to provide loading information to WASP on virtually any time scale, from timestep to timestep, to year average loads.

1.5.7. Hydrodynamics

There are currently three surface flow options available for WASP. The first two options pertain to how WASP will calculate the exchange of mass between adjoining segments with flow in both directions across a segment interface. The three flow options available for surface water flow are:

1. Gross Flows -- WASP will calculate net transport across a segment interface that has opposing flow. WASP will net the flows and move the mass from the segment that has the higher flow leaving. If the opposed flows are equal no mass is moved.
2. Net Flows -- Pertains to mass and water being moved without regard to net flow.
3. Kinematic Wave -- For one-dimensional, branching streams or rivers, kinematic wave flow routing is a simple but realistic option to drive advective transport. The kinematic wave equation calculates flow wave propagation and resulting variations in flows, volumes, depths, and velocities throughout a stream network.
4. Hydrodynamic Linkage -- Realistic simulations of unsteady transport in rivers, reservoirs, and estuaries can be accomplished by linking WASP7 to a compatible hydrodynamic simulation. This linkage is accomplished through an external "hyd" file chosen by the user at simulation time. The hydrodynamic file contains segment volumes at the beginning of each time step, and average segment interfacial flows during each time step. WASP7 uses the interfacial flows to calculate mass transport, and the volumes to calculate constituent concentrations. Segment depths and velocities may also be contained in the hydrodynamic file for use in calculating reaeration and volatilization rates. Before using hydrodynamic linkage files with WASP, a compatible hydrodynamic model must be set up for the water body and run successfully, creating a hydrodynamic linkage file with the extension of

*.hyd. This is an important step in the development of the WASP input file because the hydrodynamic linkage file contains all necessary network and flow information. When Hydrodynamic Linkage is selected in the Data Set Parameters screen, the user cannot provide any additional surface flow information. When you are ready to begin the development of a WASP input deck, simply open the hydrodynamic linkage file from within the data preprocessor. The hydrodynamic linkage dialog box allows the user to browse and select a hydrodynamic linkage file. The data preprocessor will open the hydrodynamic interface file and extract the number of segments, the starting and ending time. The data processor will also determine the set of boundary segments (segments that receive flow from outside the model network) and set the boundary concentrations to 1.0 mg/L. Once a hydrodynamic linkage file is selected in the data preprocessor, WASP has enough information to execute a simple test run with no loads or kinetics enabled. This step is recommended to test the network and transport integrity. If the simulation is run for a sufficient duration, concentrations should approach 1.0 mg/L throughout the network. If you are getting a number other than 1 mg/L, you may have to use a different time step in the hydrodynamic model. This is especially true if the concentrations are oscillating between large and small numbers, a clear indication of numerical instability. WASP has the ability to get hydrodynamic information from a host of hydrodynamic models. If a hydrodynamic model does not support the WASP linkage it is relative straightforward to create a hydrodynamic linkage file (see Appendix X for file format). The hydrodynamic models that currently support the WASP7.x file format are: EFDC (three dimensions), DYNHYD (one dimension branching), RIVMOD (one dimension no branching, CE-QUAL-RIV1 (one dimension branching), SWMM/Transport (one dimension branching, SWMM/Extran (one dimension branching)

1.5.8. Solution Technique

The user now has the ability to select the model solution technique to be used during the simulation. Currently there are 3 solution techniques that can be selected: 1) Euler – which is the traditional solution technique that has been in WASP since its inception, 2) COSMIC Flux Limiting – this solution technique is typically used when WASP is linked to multi-dimensional hydrodynamic models like EFDC, 3) Runga-Kutta 4 step solution technique used for diurnal simulations.

1.5.9. Time Step Definition

Starting with WASP Version 7.3 the user no longer has control over the computational timestep. Timestep optimization routines have been refined to the point where the model can determine what the most appropriate timestep should be used next. This assures the most efficient run time as well as minimizing numerical dispersion caused by too small of a time step. While the user can not set the time step directly, they do have some control over what would be an acceptable time step.

> Note: In WASP Version 7.30 the user can no longer specify the model time step

Fraction of Maximum Timestep

This dialog box specifies what fraction of the model calculated time step will be used for the next time step. Its primary purpose is aid the user in keeping the model stable. The default is 0.9 (or 90%) of the optimal time step.

Maximum Timestep

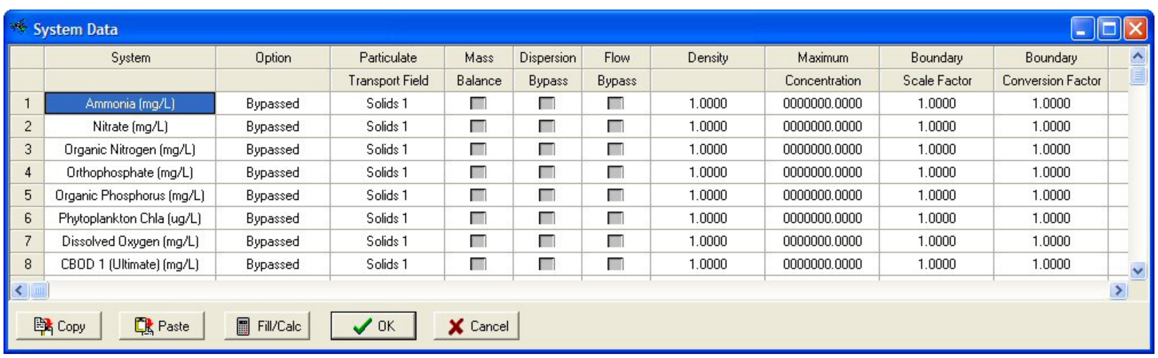
This specifies the maximum time step that will be used. If the time step optimizer calculates a time step larger than this value, this value will be used. This could be important in constraining the time step for diurnal or daily calculations.

Minimum Timestep

This specifies the maximum time step that will be used. The default minimum time step is defined in the model as 0.0001 days. Use this dialog to raise the minimum time step.

1.6. Systems

The system data entry form allows the user to define system specific information. A system in WASP is a state variable within the model. The state variables in WASP change from one model type to another. The user controls, which state variables, will be considered in their model input dataset from within this screen.



	System	Option	Particulate Transport Field	Mass Balance	Dispersion Bypass	Flow Bypass	Density	Maximum Concentration	Boundary Scale Factor	Boundary Conversion Factor
1	Ammonia (mg/L)	Bypassed	Solids 1				1.0000	0000000.0000	1.0000	1.0000
2	Nitrate (mg/L)	Bypassed	Solids 1				1.0000	0000000.0000	1.0000	1.0000
3	Organic Nitrogen (mg/L)	Bypassed	Solids 1				1.0000	0000000.0000	1.0000	1.0000
4	Orthophosphate (mg/L)	Bypassed	Solids 1				1.0000	0000000.0000	1.0000	1.0000
5	Organic Phosphorus (mg/L)	Bypassed	Solids 1				1.0000	0000000.0000	1.0000	1.0000
6	Phytoplankton Chla (ug/L)	Bypassed	Solids 1				1.0000	0000000.0000	1.0000	1.0000
7	Dissolved Oxygen (mg/L)	Bypassed	Solids 1				1.0000	0000000.0000	1.0000	1.0000
8	CBOD 1 (Ultimate) (mg/L)	Bypassed	Solids 1				1.0000	0000000.0000	1.0000	1.0000

Figure 1-8 WASP System Bypass and Global Scale Factors

1.6.1. System Options

There are three options for this field: Simulated, Constant, and Bypassed. The user can select which of the system options by selecting the option from the drop down dialog box for each individual system.

- Simulated: indicates to WASP that the user wants the model to calculate all equations associated with that state variable every time step. This is the most common selection.
- Constant: indicates to WASP that the user wants to hold the mass of this system constant and not allow the equations pertaining to this system to be calculated but allow its mass to influence the rates and fate of the other system's that can be affected by the presence of this systems mass. An example would be to include the influence of algae on dissolved oxygen without simulating the dynamics of algae. The user would provide initial concentrations for algae (that would never change), and enter rate constants for respiration and oxygen production. This would simulate a steady state effect of algal influences on dissolved oxygen without providing all the information needed to simulate algae.
- Bypassed: indicates to WASP that NO calculations should be done for the particular system. When a system is bypassed in WASP the user does not have to provide boundary concentrations or initial conditions. When bypassing systems in WASP make sure that you are not removing an integral part of the problem you are trying to solve.

For both the advective and dispersive transport functions in WASP, the user has the ability to bypass the effect of the particular transport phenomenon on the particular state variable in WASP. If the user would like to see the effect of algae on the system when it is not allowed to transport, the user sets the bypass flag for Chlorophyll-a to "Y" in either advection or dispersion (possibly both)

1.6.2. Dispersion/Flow Bypass

The dispersion/flow bypass option allows the user to specify whether a state variable will transport by either one of these processes. If the user did not want a state variable to be affected by dispersion or flow they should check the appropriate box.

1.6.3. Density

The density of each constituent must be specified under initial conditions as well (g/cm^3).

1.6.4. Maximum Concentration

The maximum concentration column allows the user to specify what would be the expected maximum concentration (mg/l) of any of the given state variables. If WASP predicted a concentration greater than the supplied value here the model simulation would be terminated.

1.6.5. Boundary/Load Scale & Conversion Factor

The boundary scale and conversion factors are specified for each individual system. The conversion factor can be used for converting the boundary time series information to the appropriate concentration units used by WASP. The scale factor can be used to attenuate the boundary concentrations without re-entering the time series data. An example would be if the user wanted to know what the effects of doubling the loads would be on water quality. Instead of re-entering the time series data, setting the scale factor to 2 would cause WASP to multiple the times series by 2.

1.7. Print Interval

The print interval is the user specified time function in which simulation results will be written to the simulation result file. The WASP model does not have to write information at every time step but can be controlled by the user. Depending on the size of the network and time frame being simulated by WASP, the simulation result files may be rather large. The user has full control over the time frame in which the information is written to the simulation result file. This function works like all other time functions in WASP. The user must provide the desired time step and simulation time that this interval is used. The user must provide at least two pairs of data.

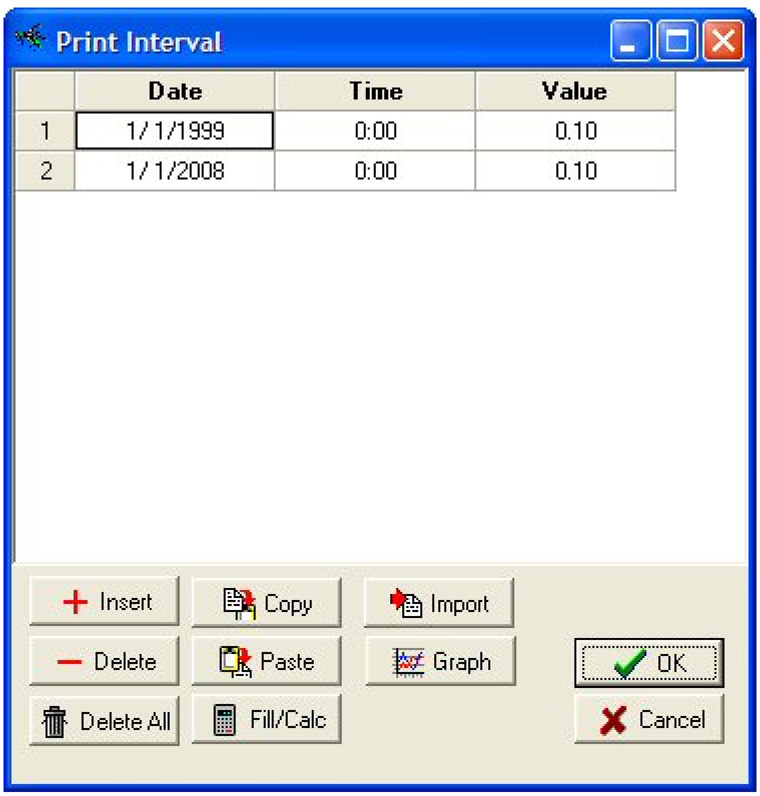


Figure 1-9 WASP Print Interval Definitions

> Note: Print Interval is a Step-wise Function

1.8. Segmentation Screen

This data entry form allows the user to define the number of segments that will be considered in the simulation. Segments are the spatial component in which WASP solves its set of equations. Segments have volume, environmental and constituent concentrations associated with them. The segment data entry form has four tables associated with them: 1) Segment Definition, 2) Environmental Parameters, 3) Initial Conditions, 4) Fraction Dissolved.

1.8.1. Segment Definition

The segment definition screen is where the user provides segment specific geometry information. It is import that the user has a good understanding in how their water body will be segmented prior to entering the information on this screen.

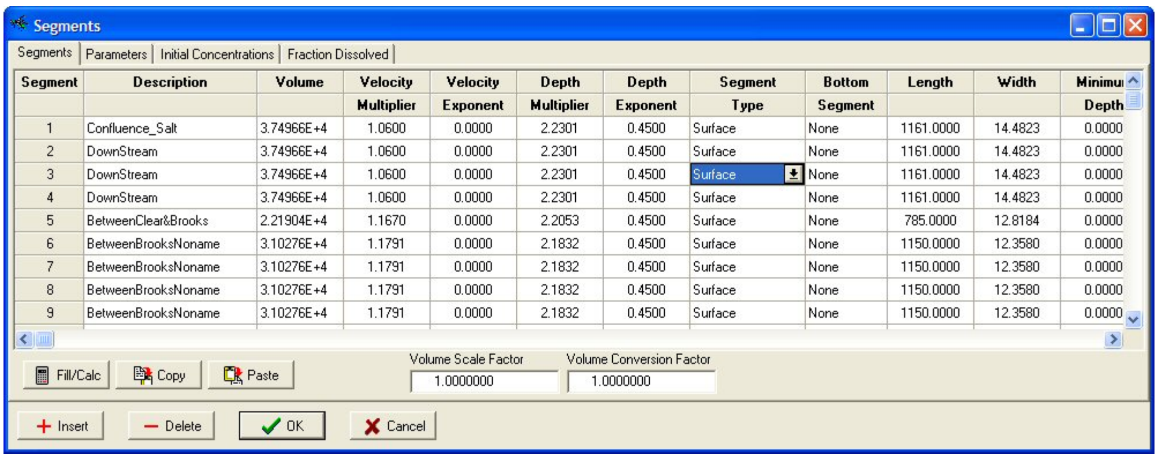


Figure 1-10 Segment Definitions

Inserting/Deleting Segments

Before the user can define a segment, the user needs to insert a segment by clicking on the insert button. This will cause a segment to be inserted at the active row in the table. If this is the first segment to be inserted it will initiate the table and insert a row at the top.

To delete a segment, highlight the row in which you want to delete and click on the delete button.

Segment Naming Convention

WASP automatically names the segments by numbers 1 through the number of segments. WASP also allows the user to give an alphanumeric name to individual segments. This alphanumeric name is there for the convenience of the user and will appear on the other screens (Dispersion, Flow) as well as in the post processor so that the user does not need to keep track of the segments by number alone. When you initially insert a segment it is automatically given the name WASP Segment. To name a segment highlight the cell and type the name for each segment, this segment name will be available in all subsequent input screens as well as model output.

> If you are pasting segment names to WASP, the names can not contain spaces or must be contained in quotes.

Volumes

This column represents the volume of the segment that is being defined. The units for volume are cubic meters. Note that WASP does not assume a cubic formation for a segment, the shape is arbitrary.

Water Velocity/Depth

There are several options for specifying water velocity and depth to WASP. Depth and velocity can be held constant by entering their values in the Depth and Velocity multiplier field and setting the exponent to zero. The user may also allow depth and velocity to vary as a function of flow. To do this the user must provide a depth and velocity multiplier and exponent. The velocity (m/s) is computed from the formulation aQ^b while the depth (m) is computed from cQ^d, where a & d are coefficients and Q is the flow (m^3/sec).

Segment Type

WASP supports four different segment types. The user must provide a segment type for each of the segments being defined. The segment type dialog box is used to define the segment type for the segment being defined.

1. Surface Water Segment – any segment that has an interface with the atmosphere. Only segment types 1's have reaeration.
2. Sub-Surface Water Segment – water segment without atmospheric interface.
3. Surface Benthic Segment – surficial benthic segment.
4. Sub-Surface Benthic Segment – all benthic segments below surface benthic segments.

Bottom Segment

The bottom segment is used to define which segment is below the currently being defined segment. If the segment does not have a segment below it, the bottom segment should be set to none or zero. The bottom segment definition is used to define the optical light path; it is not used in transport calculations.

Length

The length is specified as the distance between the upstream and downstream interface of the segment being defined. This length is used in the kinematic wave flow calculation. If the user does not specify a volume, but specifies: length, width and depth; the volume will be calculated by the model at run time.

Width

The width is specified as the average width of the segment along its length. The width is used and adjusted in the kinematic wave calculation for flow.

Minimum Depth

The minimum depth is specified as the lowest water depth a segment will become before transport (i.e. flow calculation) will cease. This is used to keep segments from drying up or draining more than they normally would.

Slope

This is the slope of the length of the segment.

Roughness

This is equivalent to the Manning's n roughness coefficient. It is used in the kinematic wave calculation.

1.8.2. Segment Environmental Parameters

This table contains segment specific environmental parameters. These parameters are different for the various WASP model types. The segment parameter information interacts directly with the Parameter Scale Factor screen.

The user only needs to provide information for the environmental parameters that are going to be considered in the simulation. Some parameters are used to directly define segment specific information (i.e. SOD); others are used to point to environmental time functions (i.e. Temperature). The pointers to environmental time functions allow the user to define spatial and temporal variation for segment parameters such as: temperature, water velocity, pH, and bacteria concentration.

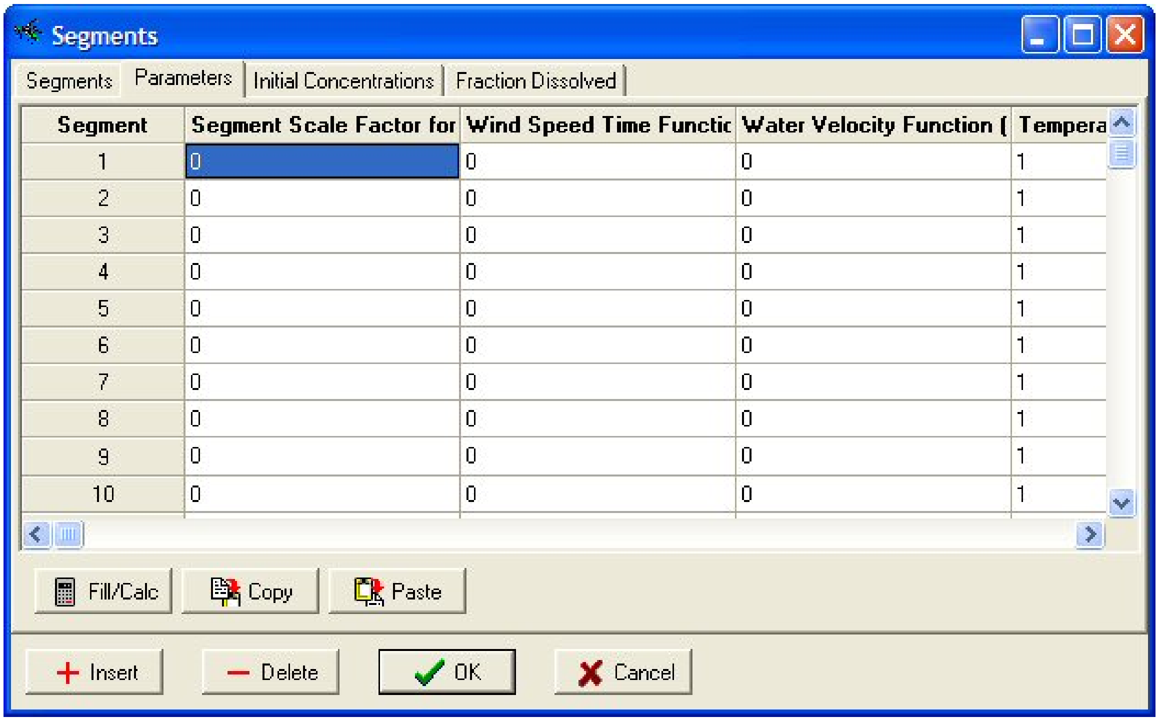


Figure 1-11 Environmental Parameters

1.8.3. Initial Concentrations

Because WASP is a dynamic model, the user must specify initial conditions for each variable in each segment. Initial conditions include the constituent concentrations at the beginning of the simulation. The products of the initial concentrations and the initial volumes give the initial constituent masses in each segment. For steady simulations, where flows and loadings are held constant and the steady-state concentration response is desired, the user may specify initial concentrations that approximate expected final concentrations. For dynamic simulations where the transient concentration response is desired, initial concentrations should reflect measured values at the beginning of the simulation.

Note: Initial Conditions can be read in using the Restart Option (Section 1.5.4)

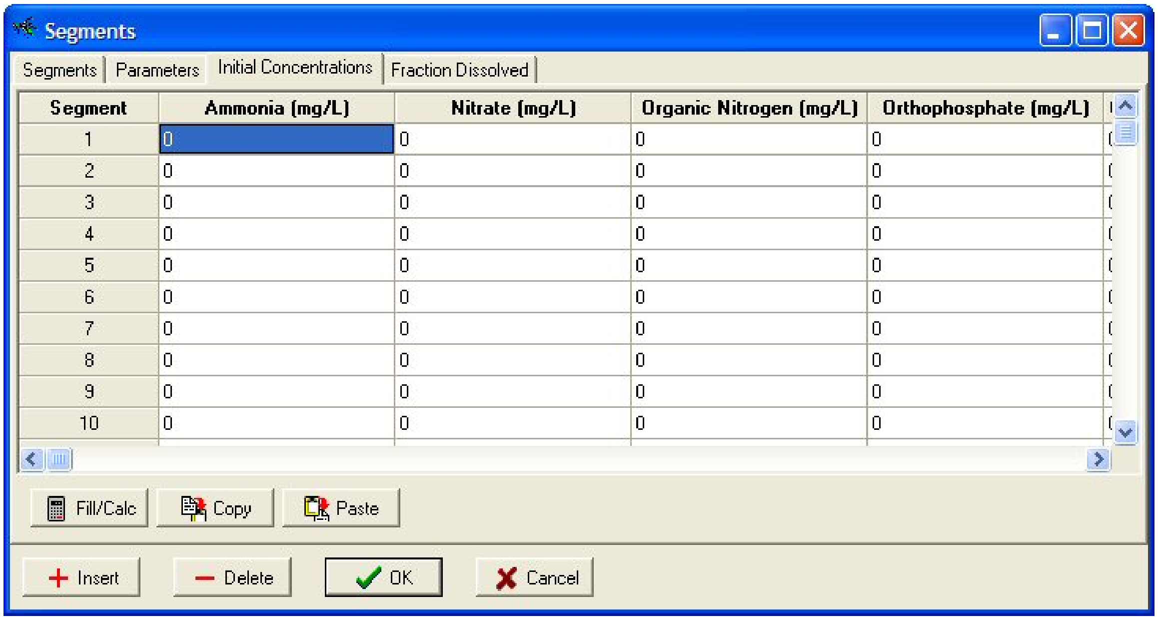


Figure 1-12 Segment Initial Concentrations

1.8.4. Fraction Dissolved

In addition to chemical concentrations, the dissolved fractions at the beginning of the simulation must be specified for each segment. For tracers, the dissolved fractions will normally be set to 1.0. For tracers, as well as dissolved oxygen, eutrophication, and sediment transport, the initial dissolved fractions remain constant throughout the simulation. For contaminants, the fraction dissolved is recomputed based upon user specified partitioning relationships.

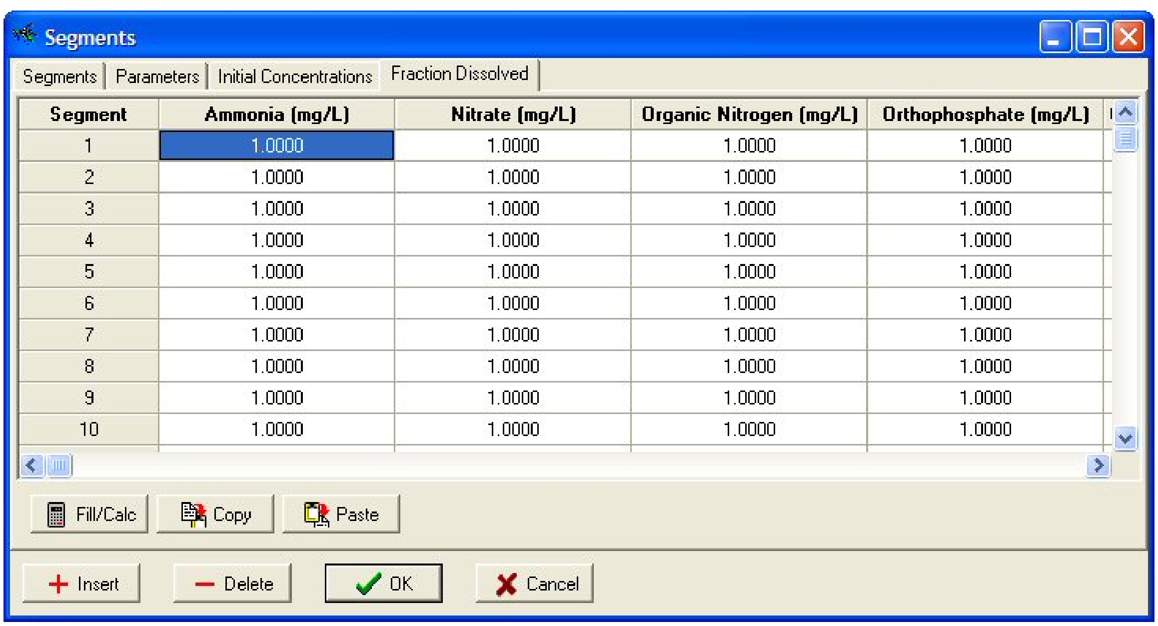


Figure 1-13 Fraction Dissolved for Constituents

1.9. Segment Parameter Scale Factors

This screen defines which parameters will be considered in the simulation as well as specifying a parameter scale factor. By default the scale factor is 1.0. Before an environmental segment parameter will be considered by WASP the used box must be checked. Un-checking this box will remove the parameter from the simulation, but all entered information is not lost. An example of using this feature is looking at the influence of SOD on dissolved oxygen. Make the first simulation with the SOD parameter checked; make the next run with it un-checked. The differences between the two runs are the influence of SOD. The user can also change the scale factors for each parameter. For example, if you wanted to double SOD set the scale factor to 2.0

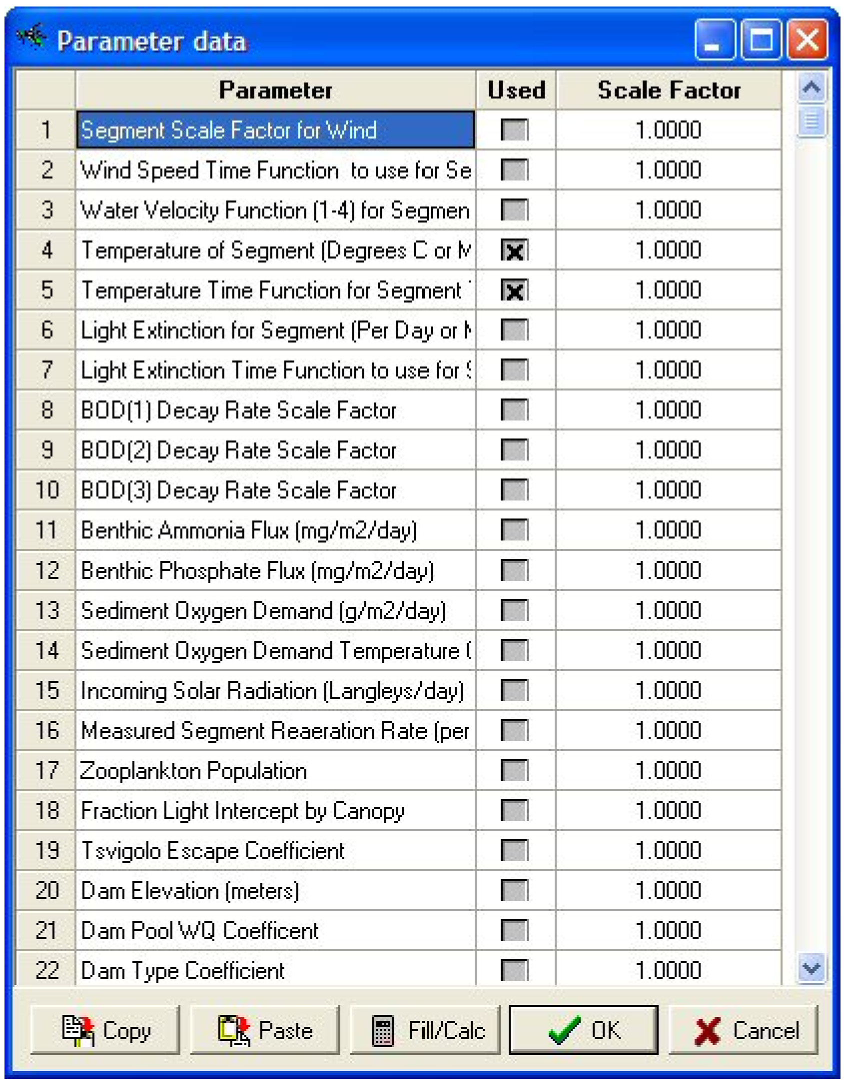


Figure 1-14 Segment Parameter Scale Factors

1.10. Dispersion

The dispersion-input screen is a complex screen that contains four tables. Under dispersion, the user has a choice of up to two exchange fields. To simulate surface water toxicant and solids dispersion, the user selects water column dispersion in the preprocessor or sets the number of exchange fields to one. To simulate exchange of dissolved toxicants within the bed, the user should also select pore water diffusion in the preprocessor or set the number of exchange fields to two.

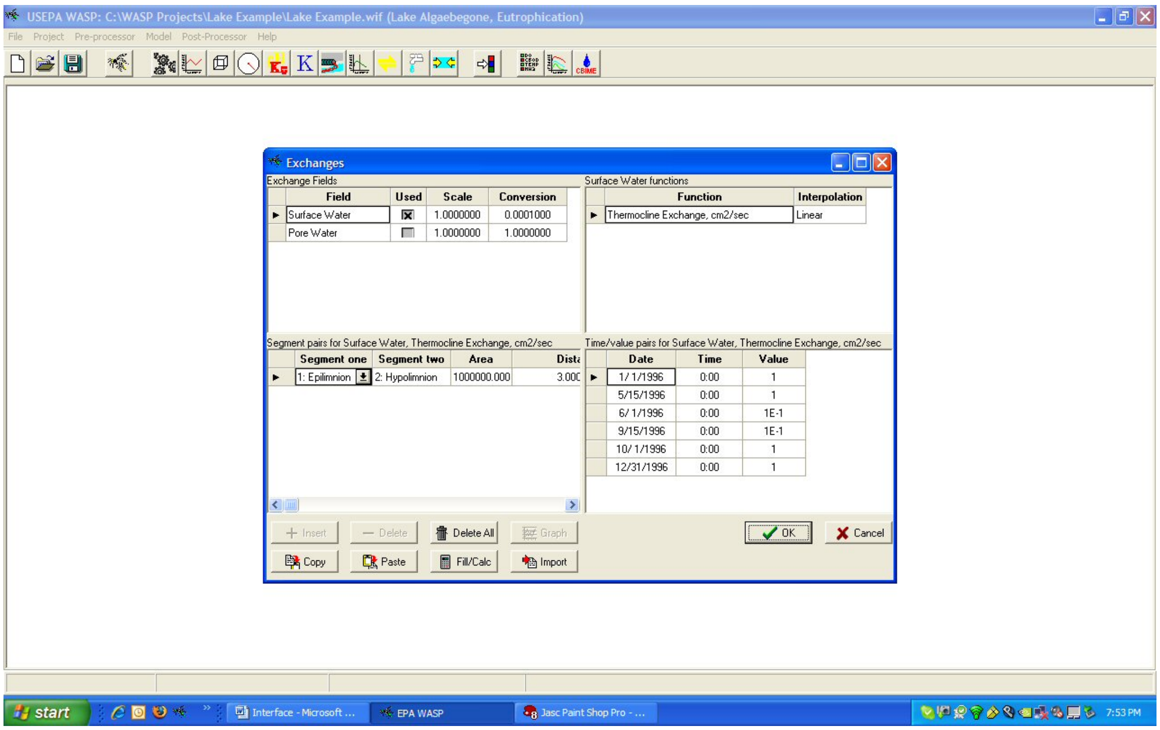


Figure 1-15 Dispersion Entry Forms

1.10.1. Exchange Fields

This table in the upper left portion of the screen allows the user to define dispersion for two types of exchanges. To use one of these exchange fields you must check the Use box and enter a scale and conversion factor. When the use box is unchecked the information for the particular exchange field is not passed to the model during execution.

1. Surface Water Exchange - The exchange of both dissolved and particulate fraction.
2. Pore Water Exchange - This exchange field moves only the dissolved portion of a constituent.

1.10.2. Dispersion Function

For each of the exchange fields the user can define several exchange functions. Each exchange function can have its own set of exchange segment pairs and a corresponding dispersion time function. WASP allows the user to provide names for each of the exchange functions. To add an exchange function click on the insert button. To delete a function, select the function by highlighting the row and click on the delete button. This will delete the corresponding segment pairs (lower left table) and the dispersion time function (lower right table).

To insert exchange functions for surface dispersion, highlight the Surface dispersion exchange field (upper left table) go over to the exchange function table (upper right table) and press insert. The bottom tables are a function of the selection in the upper tables.

Segment Pairs

The segment pairs define the between which an exchange will occur. It does not matter in which order they are defined. Neither the preprocessor, nor the model makes any checks to make sure the segments are connected in any manner. Connectivity is the responsibility of the user.

Cross Sectional Area

Cross-sectional areas are specified for each dispersion coefficient, reflecting the area through which mixing occurs. These can be surface areas for vertical exchange, such as in lakes or in the benthos. Areas are not modified during the simulation to reflect flow changes.

Characteristic Mixing Length

Mixing lengths or distance are specified for each dispersion coefficient, reflecting the characteristic length over which mixing occurs. These are typically the lengths between the center points of adjoining segments. A single segment may have three or more mixing lengths for segments adjoining longitudinally, laterally, and vertically. For surficial benthic segments connecting water column segments, the depth of the benthic layer is a more realistic mixing length than half the water depth.

1.10.3. Dispersion Time Function

Dispersive mixing coefficients can be specified between adjoining segments, or across open water boundaries. These coefficients may represent pore water diffusion in benthic segments, vertical diffusion in lakes, and lateral and longitudinal dispersion in large water bodies. Values can range from 1×10^{-10} m^2/sec for molecular diffusion to 5×10^2 m^2/sec for longitudinal mixing in some estuaries. Values are entered as a time function series of dispersion and time, in days.

1.11. Flows

The flow group works exactly the same way as the exchange group. The only difference is that the advective group has 6 transport processes that can be defined by the user.

1. Surface Water Flow – This group transports both the particulate and dissolved fractions of a constituent. If the user has selected the hydrodynamic linkage option they will not be able to enter information here.

2. Pore Water – This group only moves the dissolved fraction of a constituent.

3. Solids Transport 1 – This group moves solids field 1

4. Solids Transport 2 – This group moves solids field 2

5. Solids Transport 3 – This group moves solids field 3

6. Evaporation/Precipitation – This field adds/subtracts water only from the model network. No constituent mass is altered.

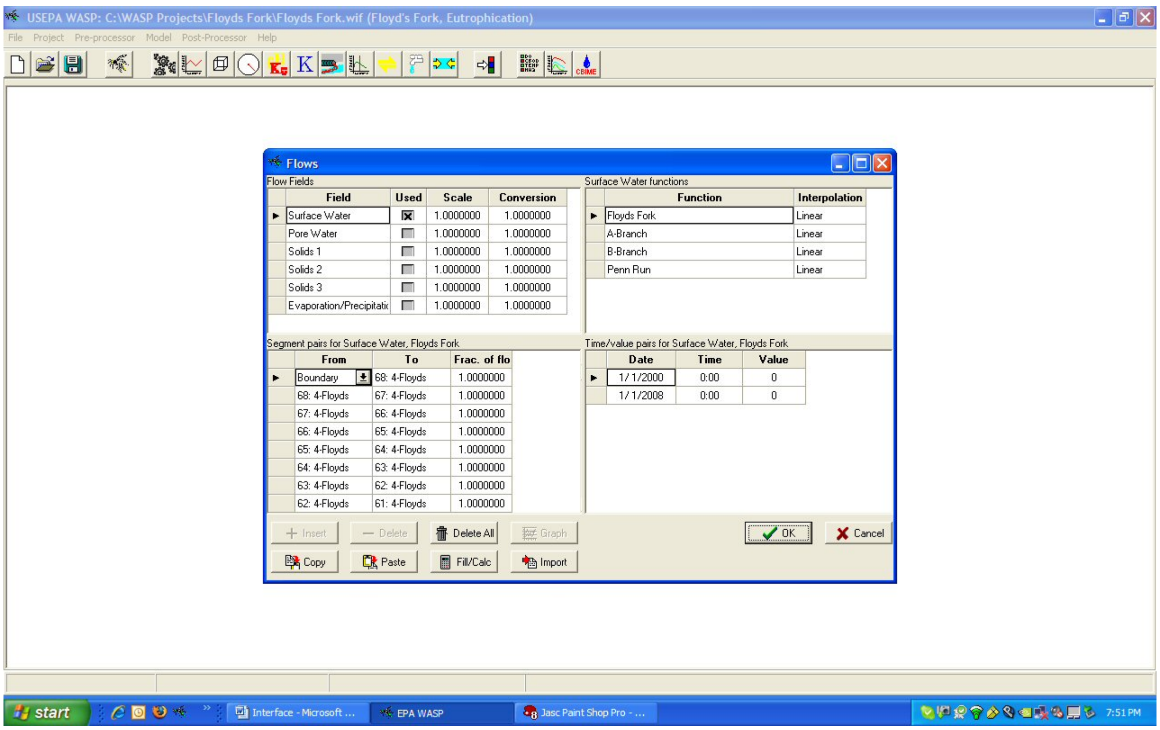


Figure 1-16 Flow Entry Forms

1.11.1. Flow Function

The user has the ability to define several low functions for each of the six flow fields. Each flow function would have its own flow continuity input (lower left table) and time variable flow input (lower right table). The user must highlight the flow field and flow function in which to enter information. WASP allows the user to provide names for each of the flow functions. To insert an exchange function click on the insert button. To delete a function, select the function by highlighting the row and click on the delete button. Note: this will delete the corresponding segment pairs (lower left table) and the flow time function (lower right table).

To insert flow functions for surface flow, highlight the Surface Flow field (upper left table) go over to the flow function table (upper right table) and press insert. The bottom tables are a function of the selection in the upper tables.

Segment Pairs

The segment pairs define the segments from/to, which flow, occurs. The order in which the segment is defined should be the path of positive flow. In other words, if segment 1 flows to segment 2, when a negative flow is entered in the time function the flow will be from 2 to 1. Note: Neither preprocessor, nor the model makes any checks to make sure the segments are connected in any manner. Connectivity is the responsibility of the user.

Fraction of Flow

The fraction of flow column allows the user to specify the fraction of the flow that transports from one to segment to the other. This field is used to split flows (diverge) for various reasons.

1.11.2. Flow Time Function

The time function table allows the user to enter time variable flow information. The user must provide the date, time and flow (cms). The user can enter the information by hand, paste in from a spreadsheet or query in from database/spreadsheets (Section 1.18).

1.12. Boundaries

Boundary concentrations must be specified for any segment receiving flow inputs, outputs, or exchanges from outside the model network. The boundary segments are automatically determined by WASP when the user defined the transport patterns. Therefore, the user cannot enter boundary information until the transport information has been entered. WASP requires that a boundary concentration be specified for every system that is being simulated for every boundary segment. To specify a boundary for a system, move the cursor to the system that a boundary needs to be specified and right click on the system.

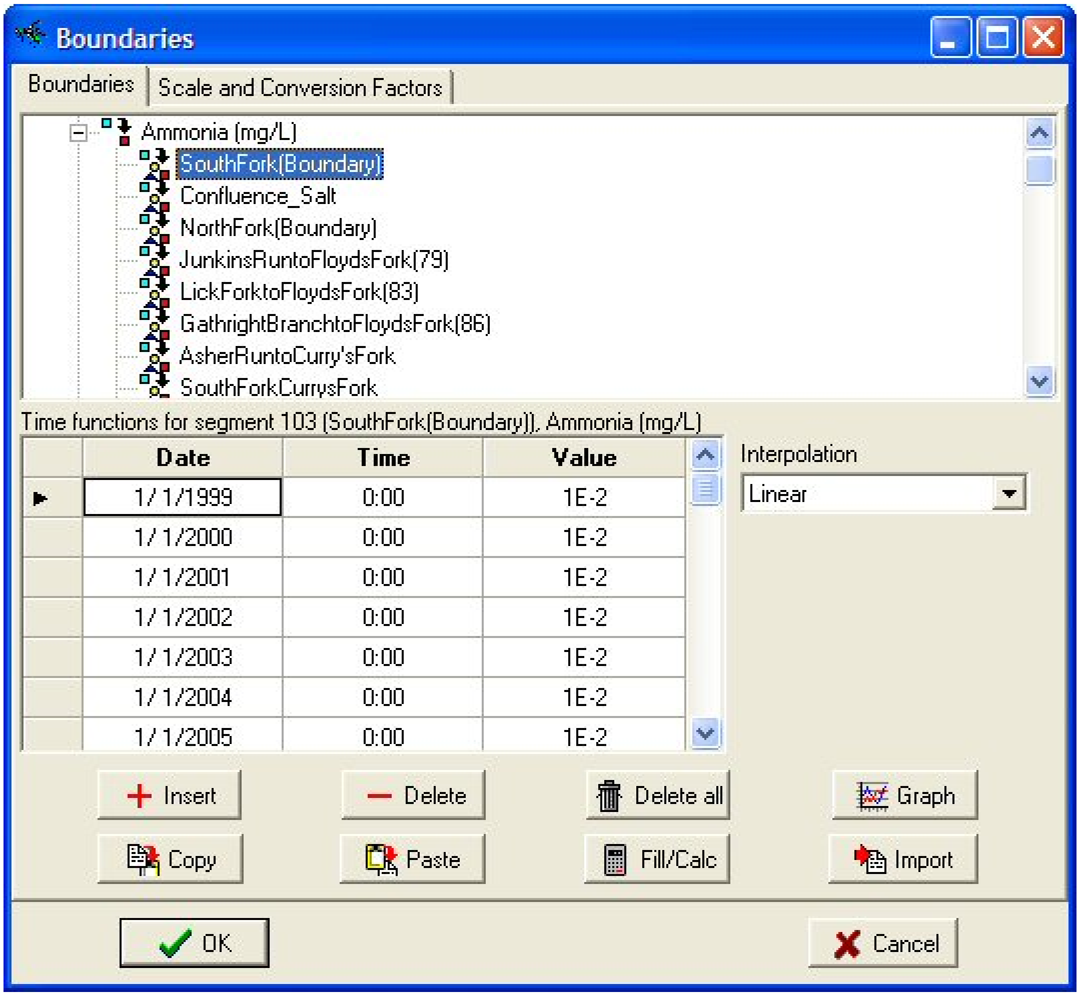


Figure 1-17 Boundary Concentration Definitions

1.12.1. Boundary Time Function

The time function table allows the user to enter time variable boundary concentrations (mg/l). The user must provide the date, time and concentration. The user can enter the information by hand, paste in from a spreadsheet or query in from database/spreadsheets (Section 1.18).

> Note: For chlorophyll-a boundary conditions the units are цg/l

1.13. Loads

Waste loads may be entered into WASP for each of the systems for a given segment. To add a load right mouse click on the system, select add load and check the segments that will be receiving a load for the selected system. Once this is done, the user will be able to select the segment to define the load. There will be an entry for every segment in which the user wants to define a load. The user can delete a load by selecting the system, right mouse click and select delete.

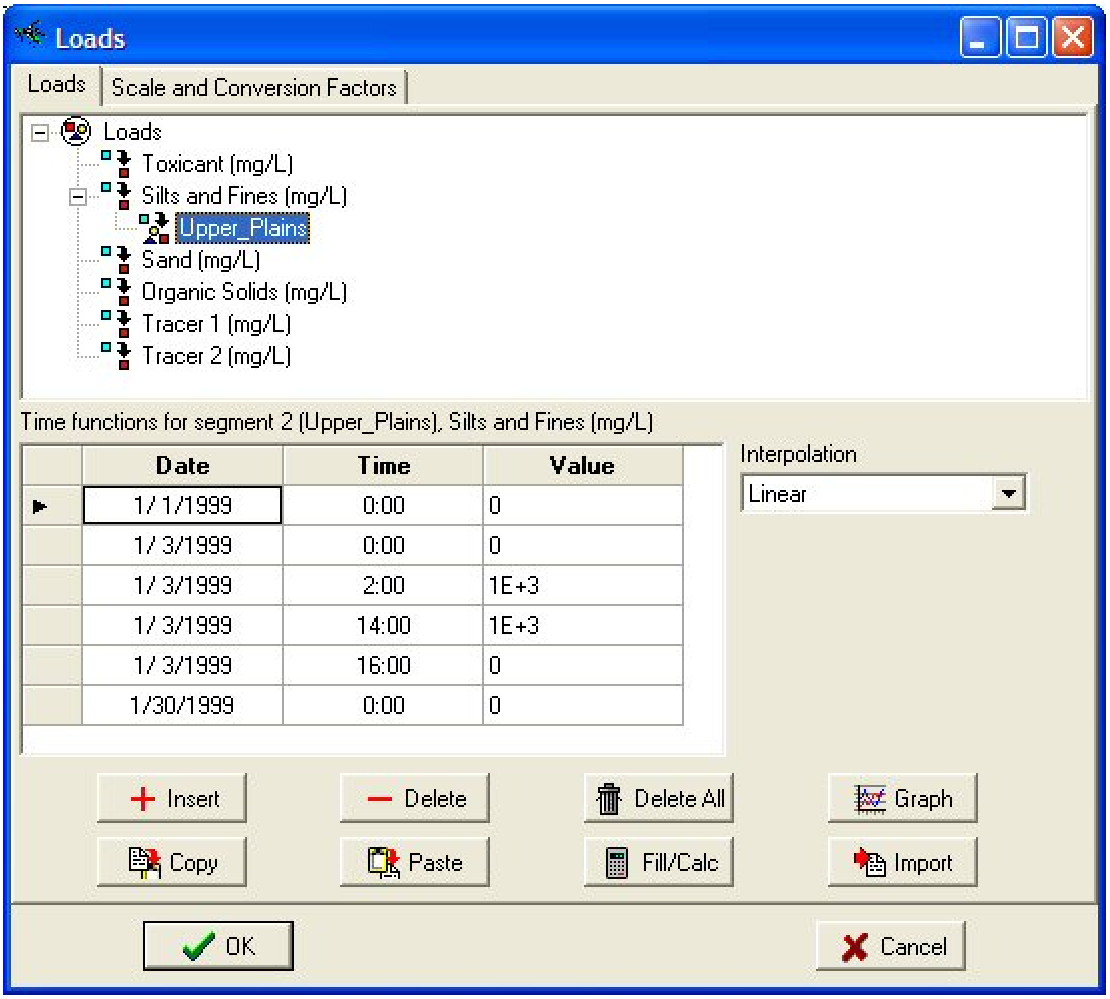


Figure 1-18 Waste Load Definition Screen

1.13.1. Load Time Function

The time function table allows the user to enter time variable loadings (kg/day). The user must provide the date, time and concentration. The user can enter the information by hand, paste in from a spreadsheet or query in from database/spreadsheets (Section 1.18).

1.13.2. Loads Scale and Conversion

The user has the ability to provide scale and conversion factors that can be used to attenuate or convert loading mass. The conversion factor for a given system can be used to convert loads measured and reported in one unit to convert to WASP required units of kg/day. The scale factor column can be used to attenuate the loads without re-entering the time function information. If the user wanted to see the impacts of doubling the loads, a scale factor of 2 would be entered for the desired system.

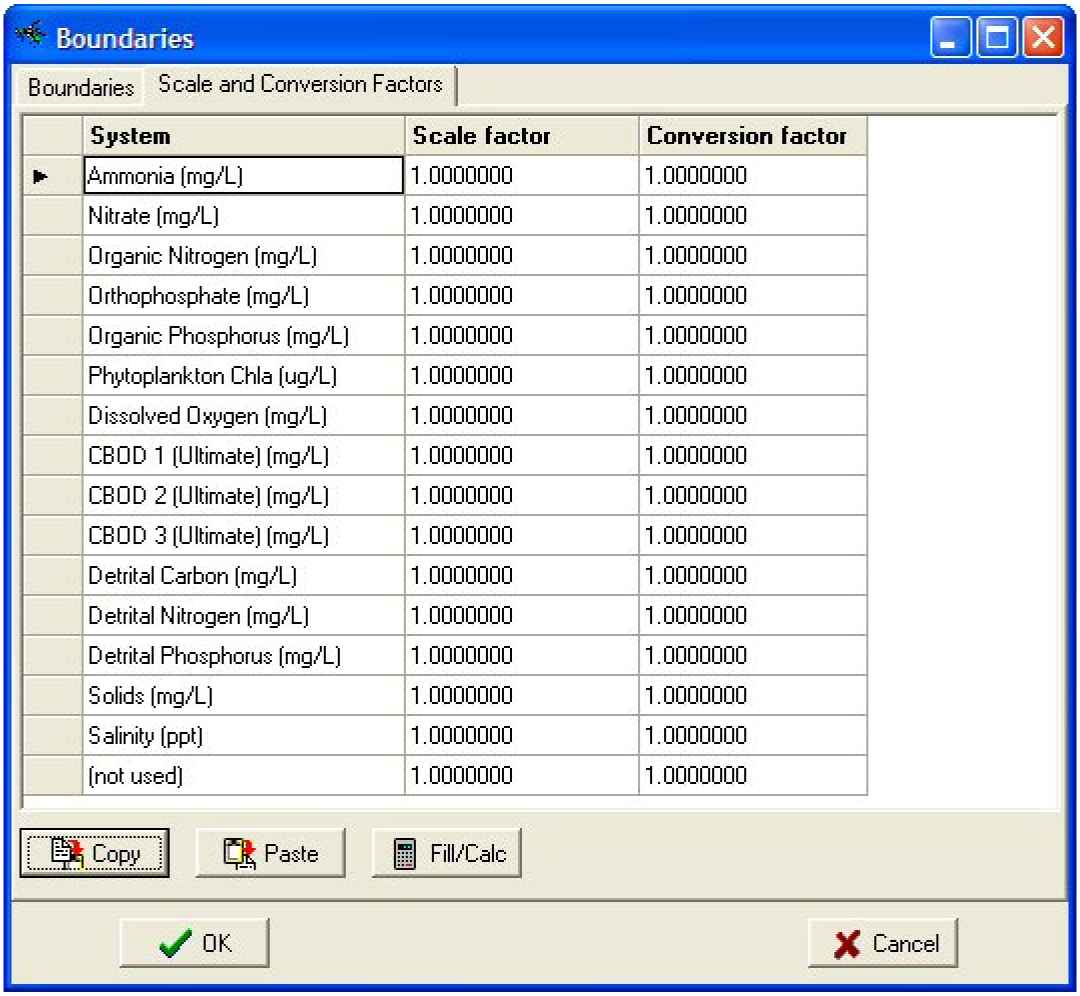


Figure 1-19 Waste Load Scale and Conversion Factors

1.14. Time Functions

The time function data entry forms allow the user to enter time variable environmental information. WASP offers a selection of all the environmental time functions for a given model type.

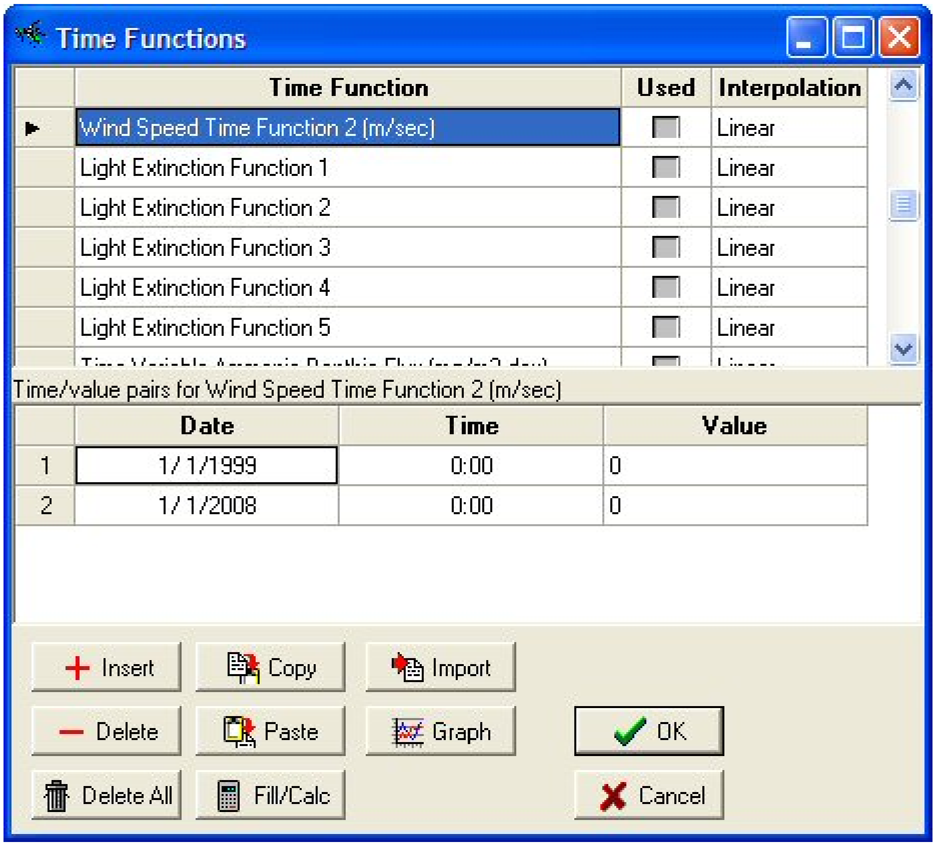


Figure 1-20 WASP Environmental Time Function Definitions

The user may provide information for all the time functions or toggle on/off any of the functions by clicking the Use dialog box. To enter information for a time function, place the cursor on the desired function. The time series data form for the given time function is displayed in the lower table. The user should enter time/date and value for the time function. The user can enter the information by hand, paste in from a spreadsheet or query in from database/spreadsheets (Section 1.18).

1.15. Constants

This data entry group includes constants and kinetics for the water quality constituents being simulated by the particular WASP model. Specified values for constants apply over the entire network for the whole simulation. The user selects which constant group they would like to define kinetic constants. To select a Constant Group the user should click on the drop-down menu for a complete list.

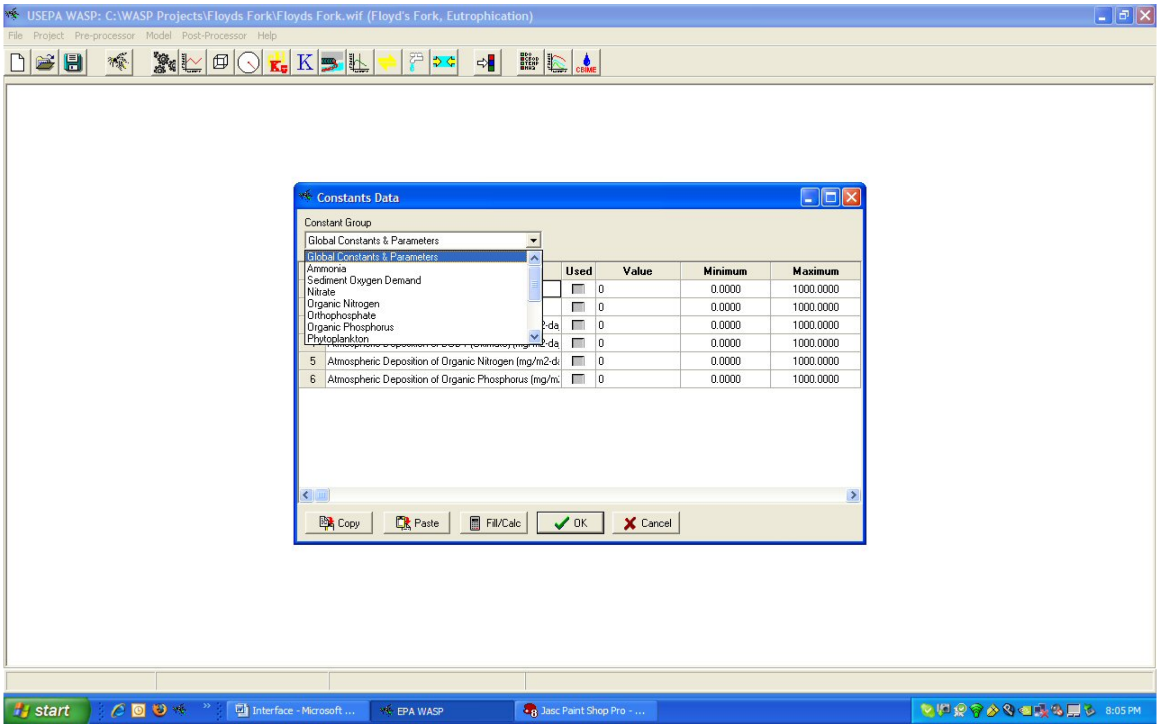


Figure 1-21 Kinetic Constant Group Selections

Once a constant group has been selected, the user is given the opportunity to enter constant data. WASP allows the user to activate constants by checking the Use dialog box and then entering a kinetic constant value. When a constant is un-checked the information is not passed onto the model, but the users' constant value is preserved.

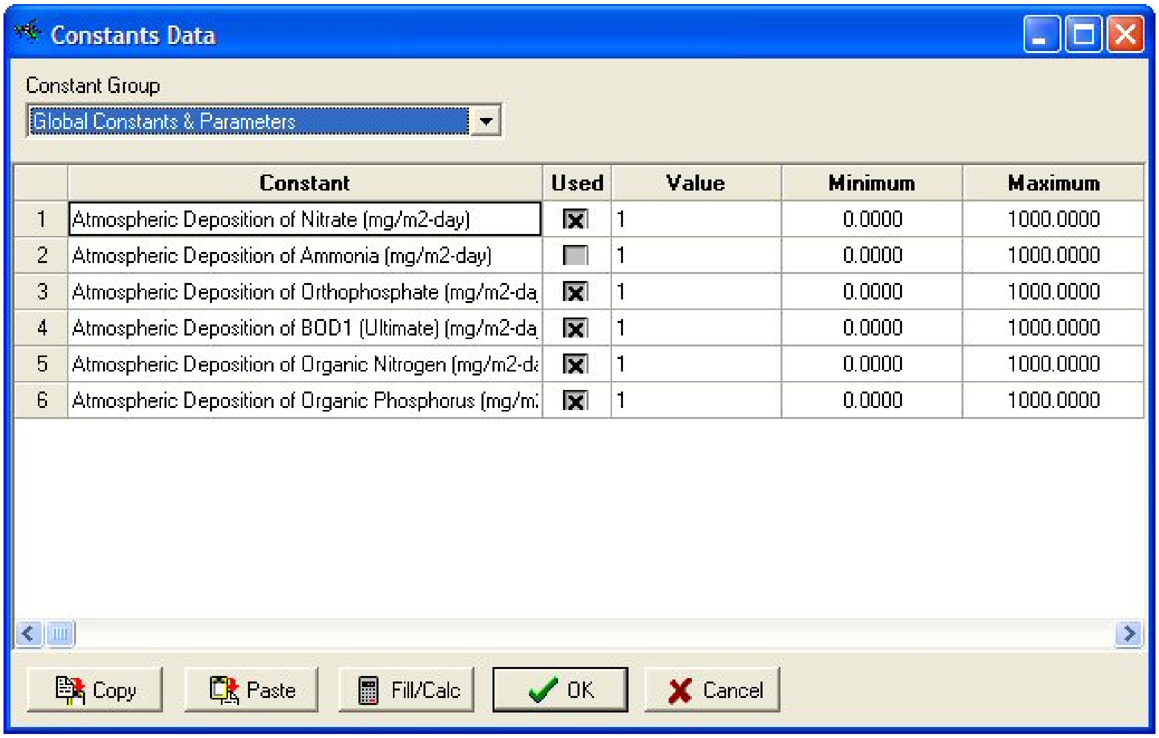


Figure 1-22 Kinetic Constant Definitions

1.16. Fill/Calculate & Graphing

Most of the data entry screens have the ability to automatically fill and make calculations on the fields of the table. To accomplish this marks the fields using standard Windows functions and then press the fill/calculate button.

To fill a column(s) just enter the value and click on okay.

To make a calculation use the variable x followed by the mathematical expression (i.e. x*2, will multiple the value on each row selected by 2)

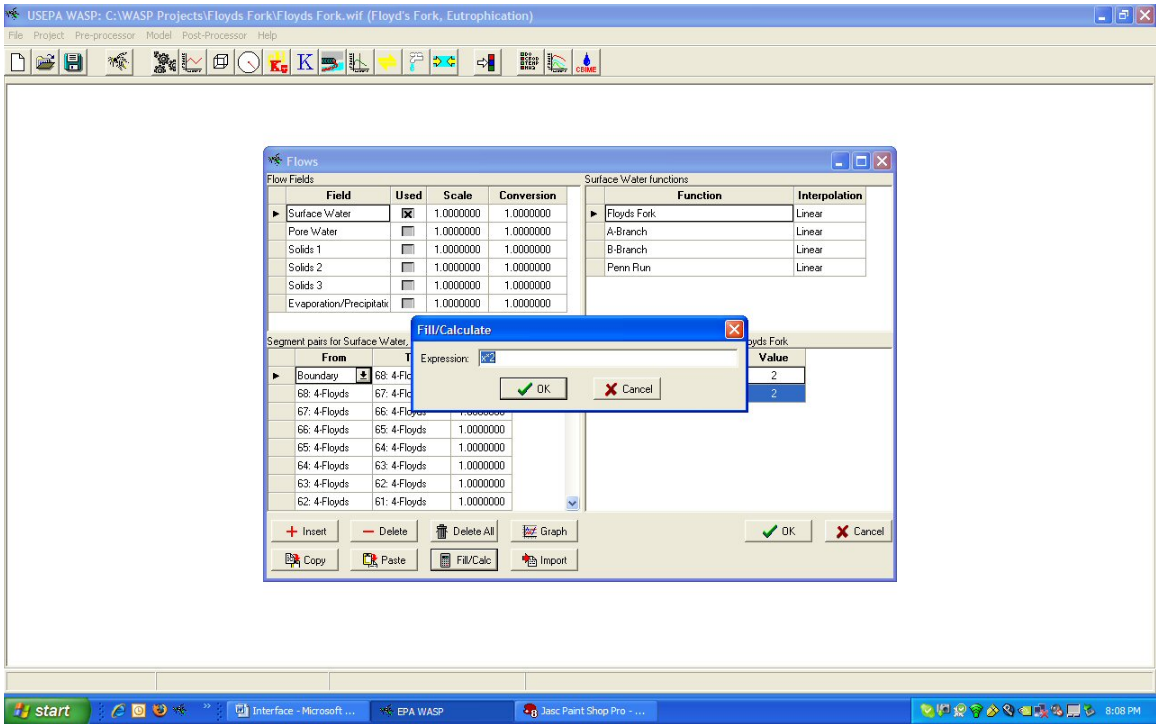


Figure 1-23 Column Fill/Calculate Option

WASP also allows the user to plot time series data from any of the appropriate tables. To plot a time series press the plot button.

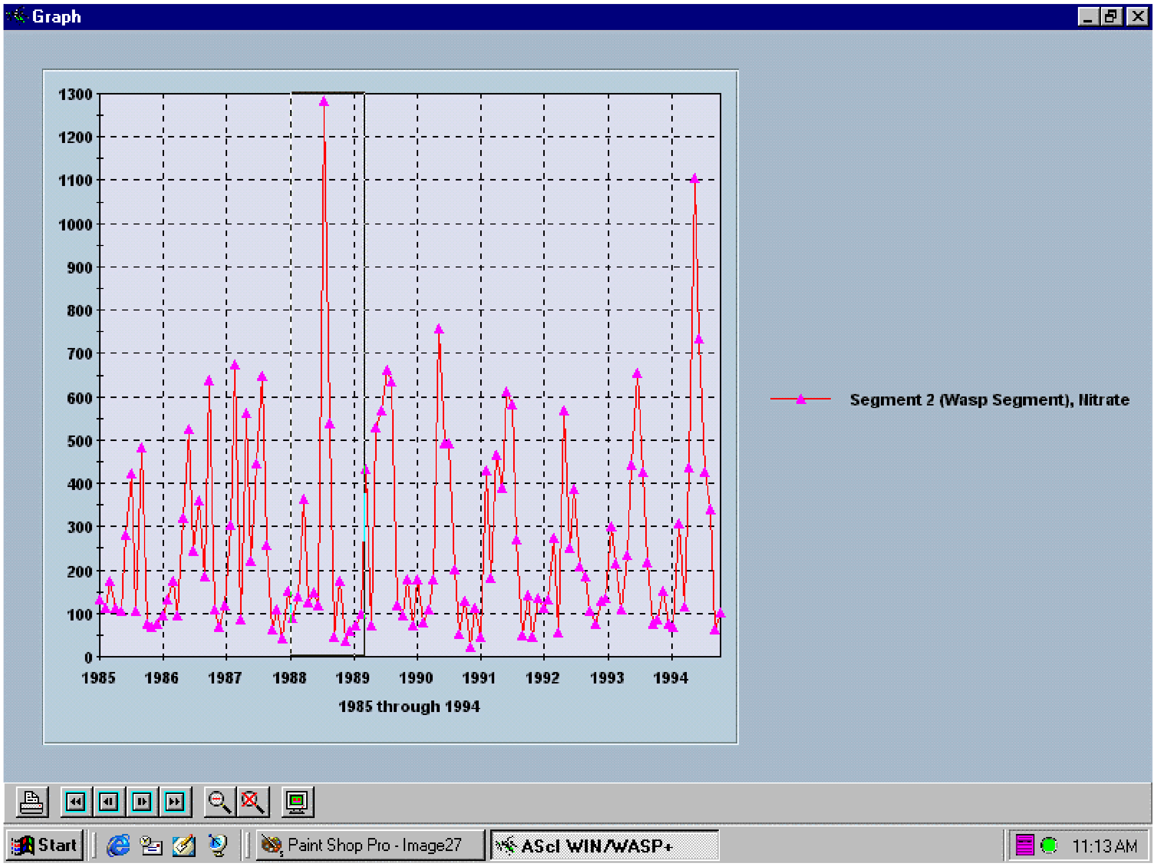


Figure 1-24 Time Series Graphing Option

The user can zoom the x and y-axis. To zoom the x-axis the user should place the cursor at the starting point of the zoom, hold down the left mouse button and drag the box to the right to select the full area to zoom. Zooming the y-axis is done the same way except using the right mouse and dragging down.

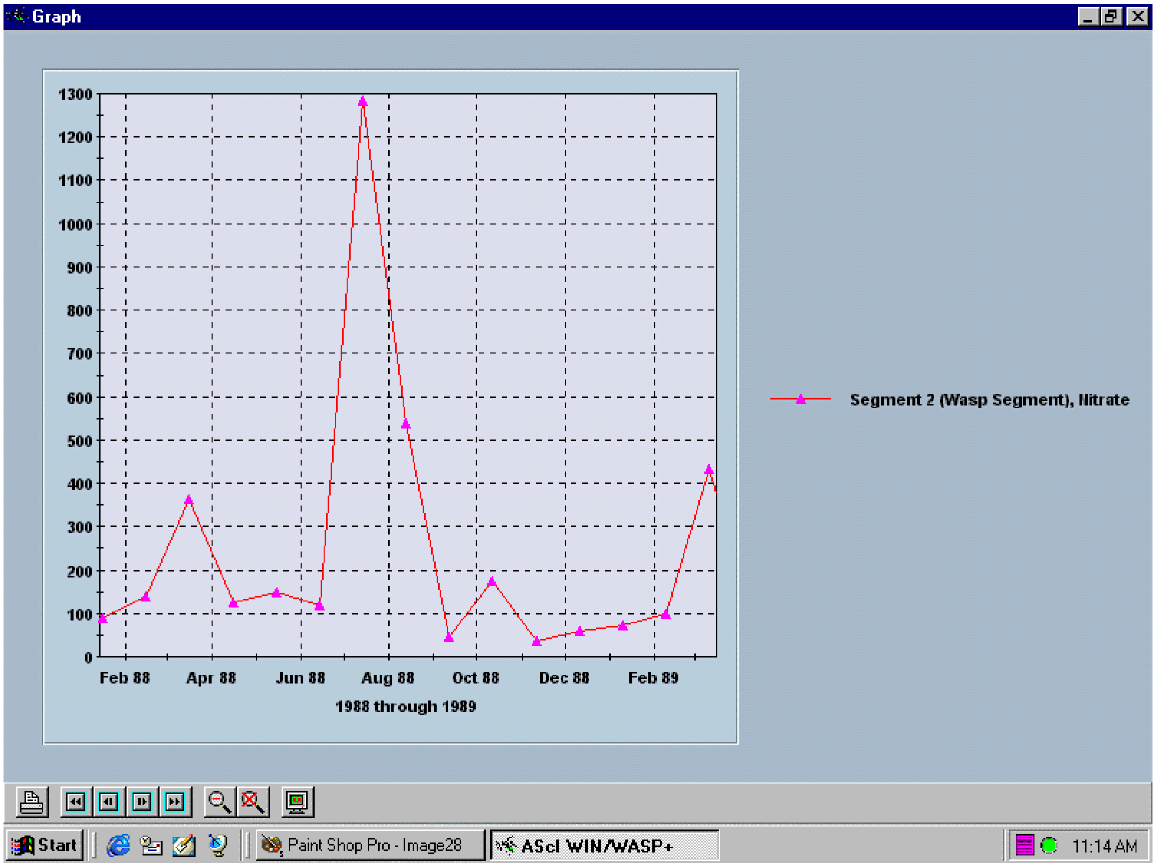


Figure 1-25 Graphing Zoom Option

1.16.1. Toolbar Definition

The user is provided a toolbar at the bottom left hand corner of the graph window. This toolbar provides basic control over the graph.

 Prints the current graph to installed printer

 Pan Zoomed Graph all the way left

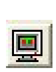
 Pan left one frame

 Pan right one frame

 Pan Zoomed Graph all the way Right

 Zoom Out One Level

 Zoom to Full Extent

 Switch Graph between Color and B&W

1.17. Model Execution

To execute the loaded input dataset the user should press the Model Execution icon on the main toolbar. WASP loads the appropriate model DLL based upon the model type set by the user in the Model Parameterization entry form.

> Note: Before you can run the model you must have an input dataset open in WASP

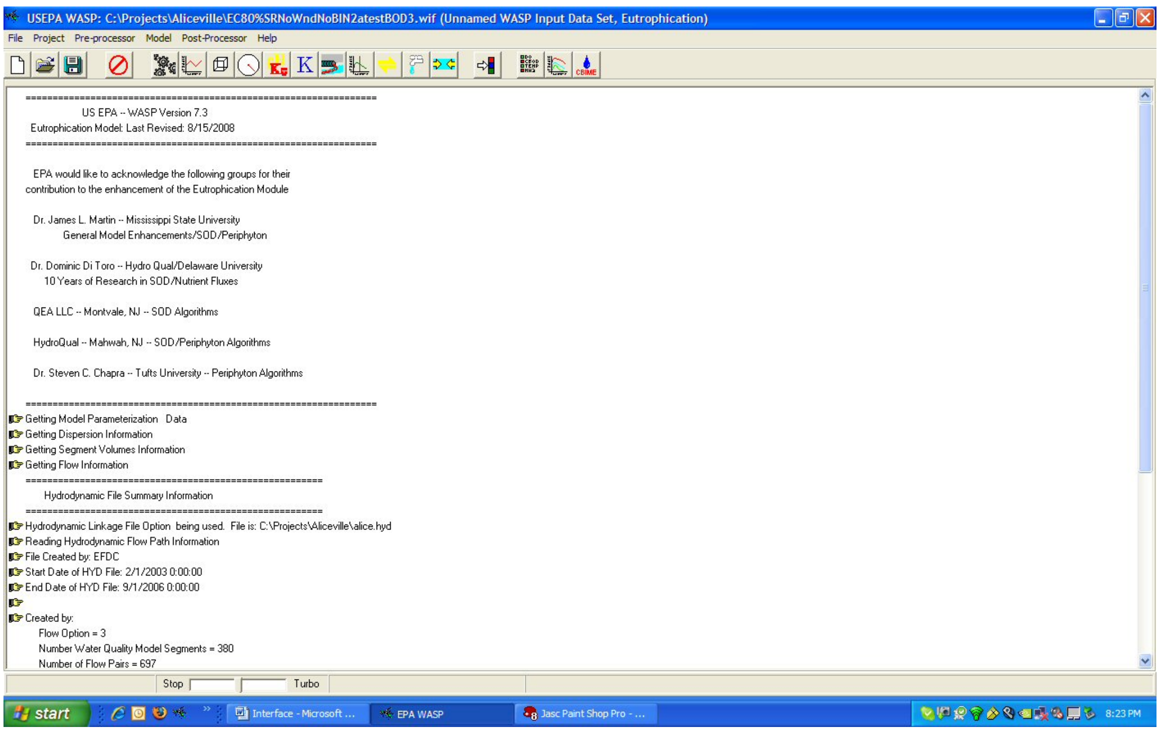


Figure 1-26 Model Data Retrieval

Once the model is executed WASP provides information back to the user on where it is in the simulation. The first set of information is the status of the data retrieval from the preprocessor. WASP does not read the conventional input files from the previous versions of WASP and WASP; it makes requests to the preprocessor for the information as it is needed. Depending upon the size of your model network and amount of time variable data this set can take some time. Once the model data has been retrieved it will begin the simulation. Once the simulation has started a grid will appear on the screen, this grid contains intermediate results for each of the state variables for each of the segments. The user can scroll this grid to look at the results. The user can shrink or stretch a column by dragging the column boundary in/out.

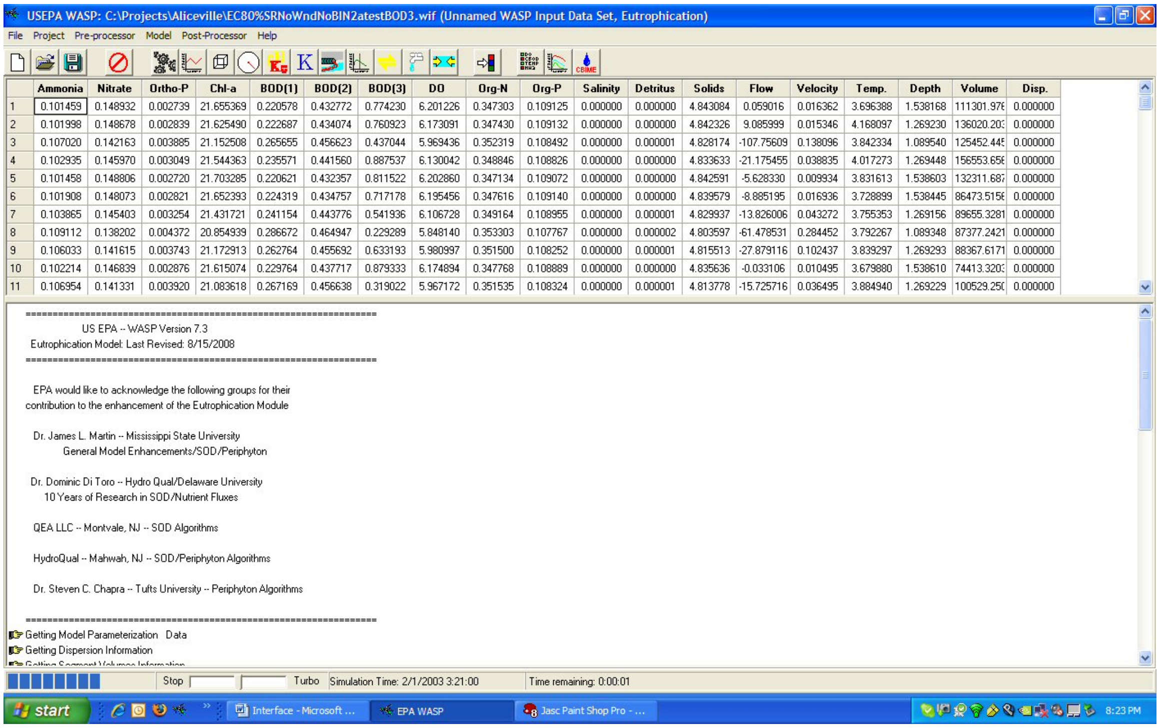


Figure 1-27 WASP Runtime Grid

1.18. Linking to External Data Source

Another major enhancement to the user interface is to allow the user to query/retrieve information from external data sources. These external data sources can be things such as database files (Paradox, Dbase, Access, etc.) or spreadsheet files (Excel). The user interface allows the user to attach to these external files and bring the time series data into the model without using the conventional cut and paste methods. Furthermore, using a graphical user interface the user has the ability to develop SQL queries that are used to subset the external file to get to the particular data that is needed for the time series being defined in WASP. If the user is working with very large datasets, the user can set a radio button that will cause the interface to execute the query steps one by one “drilling” or sub-setting the data make it more efficient to work with. Figure 1-28 illustrates a typically time series entry screen (WASP Boundary Conditions), to initiate an import the user needs to click on the Import Button.

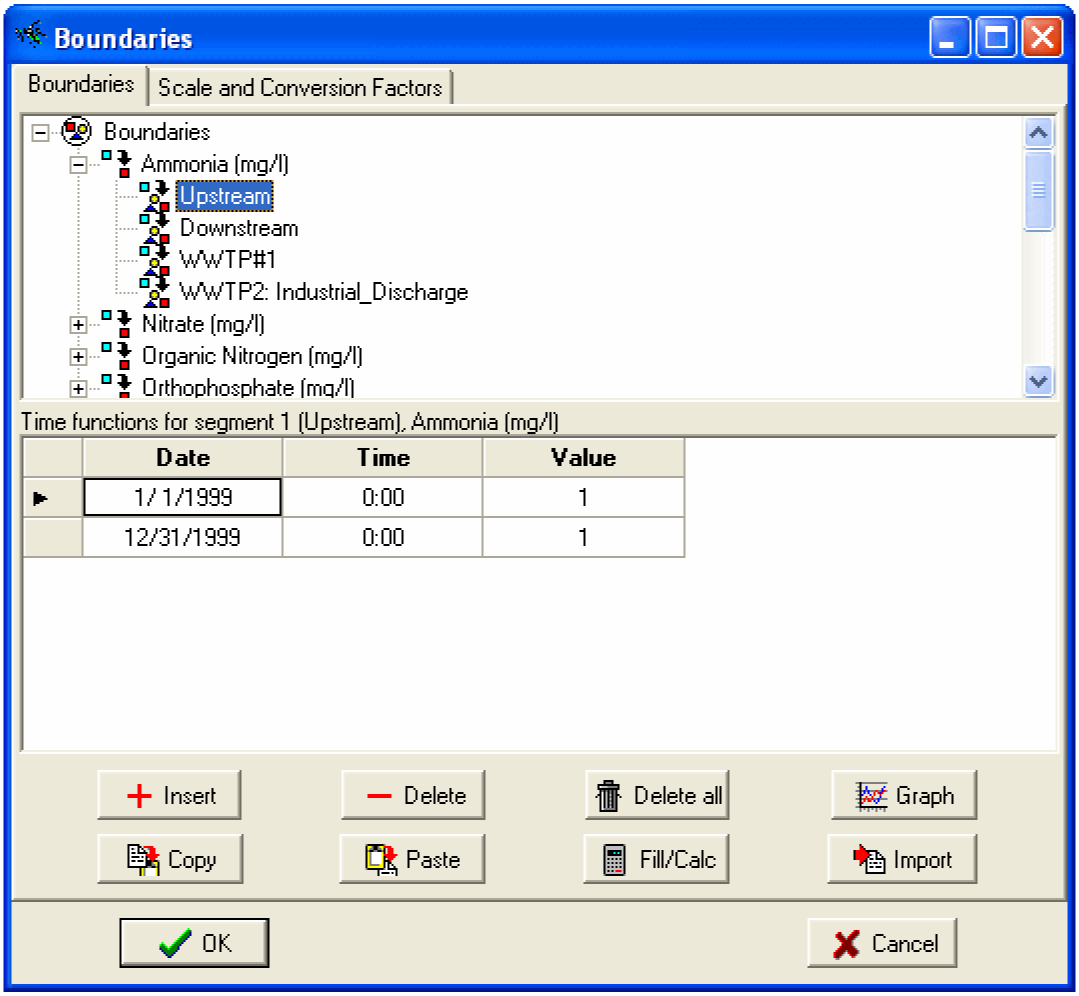


Figure 1-28 Time Series Import from External Files

Once the user selects import time series a data query form is presented (Figure 1-29). The user can select various file types in which to import data. The user should use the browse button to open the file selection dialog box, the path and filename is stored in the WIF file once selected and saved.

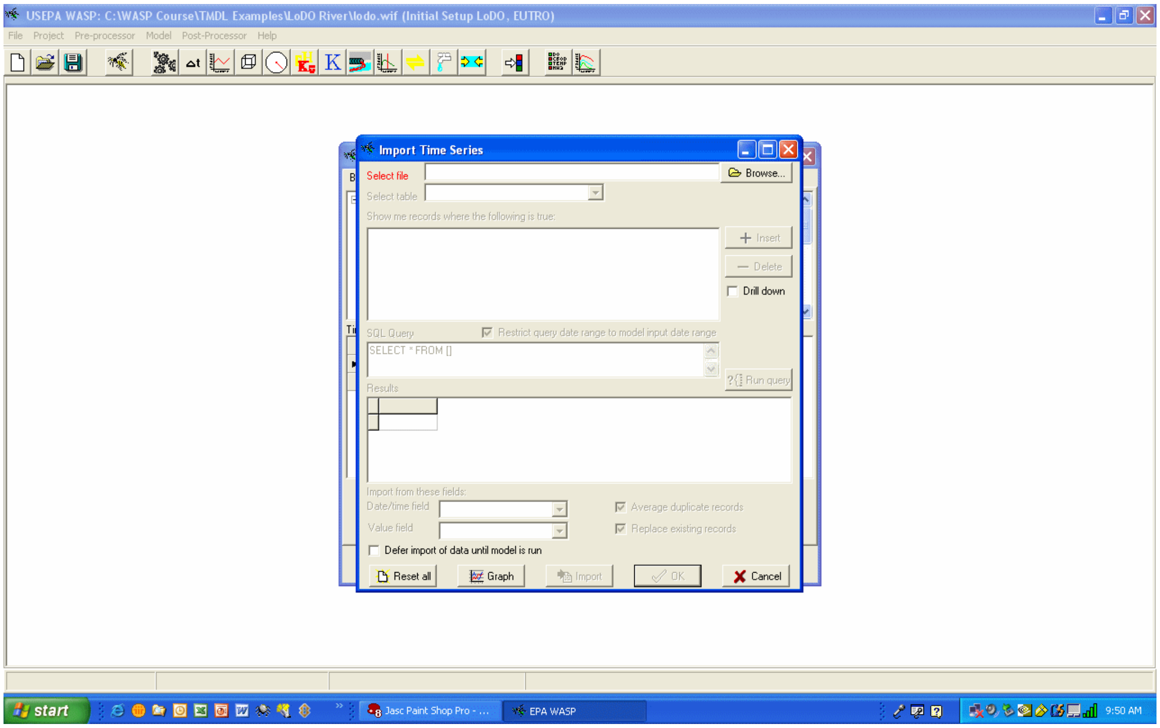


Figure 1-29 Import Time Series Control Screen

1.18.1. Sources

From the file dialog window the user will be presented with a typical windows file selection interface. The user can set the file type filter by using the drop down menu at the bottom of the dialog box. This also specifies the types of external data file types available to the user. A list of the file types are described below:

1. DBF – this is a conventional dBase file format. dBase files are standard with ESRI products and many third party software packages support this file format. Spreadsheet programs like Excel, Lotus 123 and Quattro have the ability to create these files.
2. XLS – this is a standard Microsoft Excel spreadsheet file. WASP can handle a spreadsheet with many sheets associated with the data. It is best to arrange the data that you want to pull into WASP in a columnwise fashion.
3. MDB – this is a standard Microsoft Access database file. WASP can handle Access database files that have multiple tables within the MDB. The user will have the ability to select which table to extract data.
4. DB – this is a standard Corel Paradox database file format. This is the same file type that the graphical post processor MOVEM reads to obtain observed data.

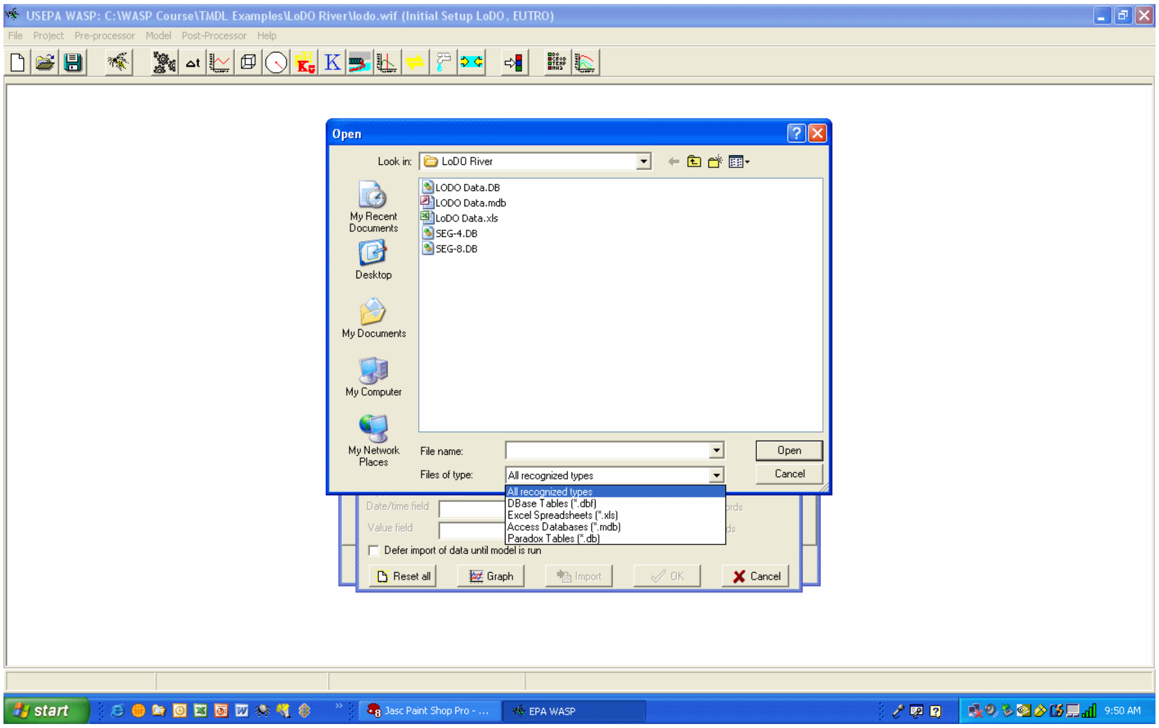


Figure 1-30 Time Series External Source Selection Dialog

If the source file is either an Excel spreadsheet file or an Access database file the user will have to select which worksheet (Excel) or table (Access) that they want to extract data. The data that is to be extracted for the particular time function must be present in a single table or worksheet, there is no provision for combining across multiple sheets or tables. The user is presented with a pick list of the available worksheets and tables available within the user selected file (Figure 1-31). For other file types the user will not be given the option for selecting anything in this pick list. For Excel spreadsheets the worksheet names that are displayed to the user are based upon the worksheet name assigned by either Excel or the user and are displayed in the tab at the bottom of the worksheet window. Furthermore, typically worksheet names when they are displayed in WASP have a "$" appended to the end of the name. For Access database files the user will be given a pick list of all of the tables and queries that are stored in the MDB file.

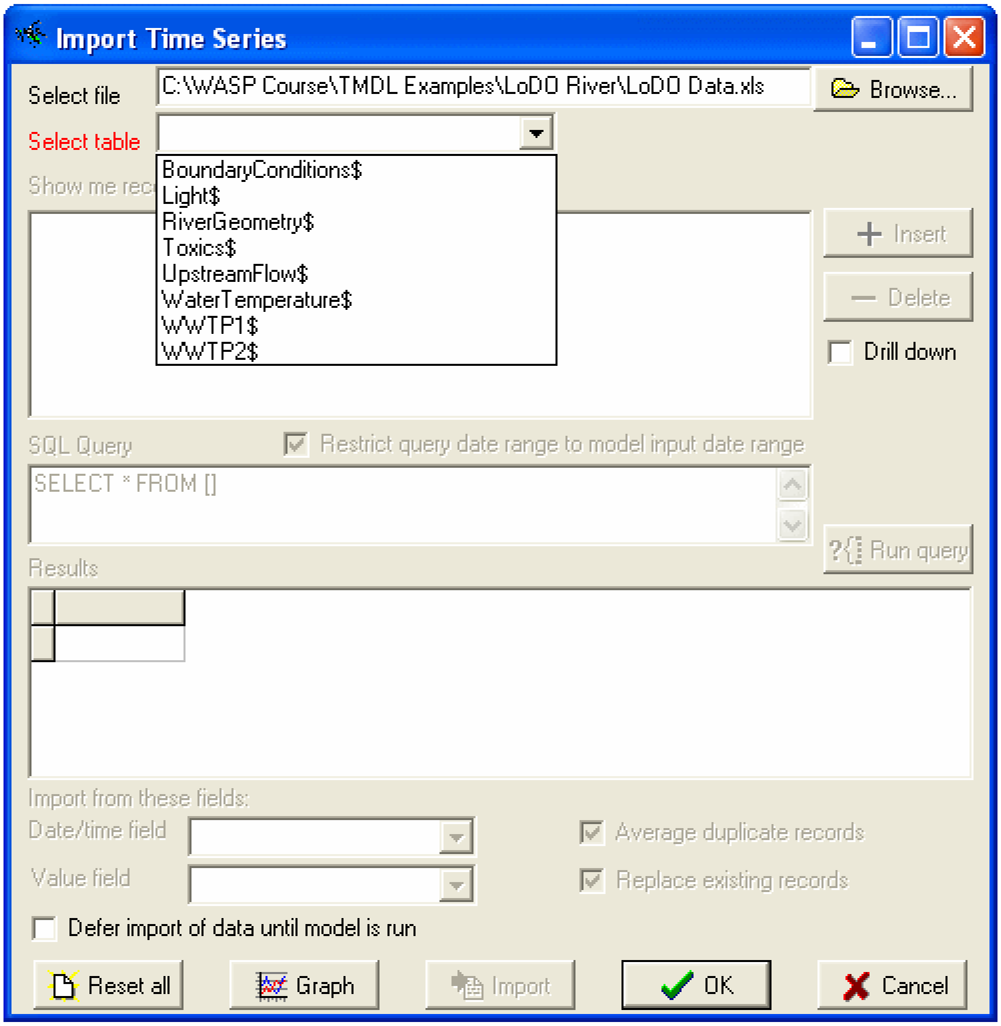


Figure 1-31 External Worksheet or Database File Selection

An example of an Excel spreadsheet file that the user can setup to extract data from is given in Figure 1-32. It should be noted that the data is arranged in a columnwise fashion with each column having a unique name. The user should take care in naming the columns so that when you are in the import function that the data is readily recognizable. If WASP analyzes the data table and some column headings are not unique it will append 1, 2, 3 …. to the end of the heading name.

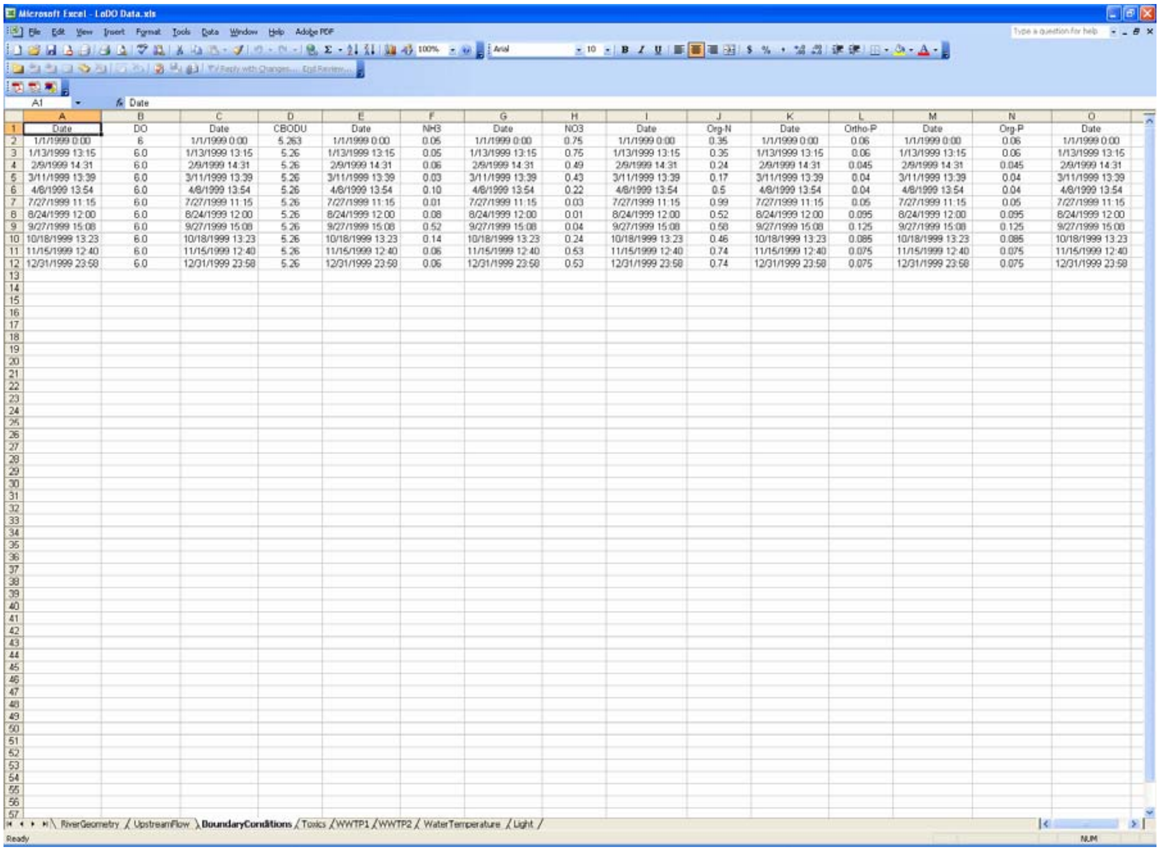


Figure 1-32 Example Excel Spreadsheet Example for External Data Source

Once the user has selected the file and tables/worksheets in which data can be extracted from the user needs to build a query to retrieve the data. For spreadsheet files because you will be typically selecting time and values (2 columns of data) you will not have to build a query, but for selecting data from a database a query will be needed to subset the data that you will be pulling into WASP. Figure 1-33 illustrates the query grid that user will user. The user needs to insert (click on Insert Button) and then define using the drop down list the field from the database that you want to set a filter for extraction. In the example the first filter is for Pcode which is a field in the database that was selected. WASP automatically builds a list of valid fields in which queries can be run from the selected files. The database has many records with different Pcodes associated with them. In this example we want to extract all of the Pcodes that are equal to NH4. The second filter a field called Station_ID was selected, this too is a field in the database, and we want to select all of the records that have a Station_ID equal to LODO Seg 4. What this query will do is select all of the ammonia timeseries data that was collected at the LODO Seg 4 station. There is a checkbox the user can select to add a time constraint on the query based upon the time domain set for the simulation prior to coming into these input screens. It is import to note that is criteria that are set here is used to subset the database for convenience and efficient extraction. What is defined in the SQL query grid is not necessarily what is going to be pulled into WASP.

If the user is working with large databases they may elect to check the Drill Down box. What drill down does is dynamically subset the database by the user defined SQL query statements. As an example, say you have a large database open for querying, you insert a query statement in for Station_ID, once you define this criteria the database is dynamically drilled down to only the records that meet the criteria for Station_ID. When you insert another query statement for Pcodes, you will only be given a list of Pcodes that are available for the defined Station_ID. Furthermore, it will develop this list much quicker as WASP does not need to go through the whole database but only the subset for the defined Station_Id.

Once the query information is defined you have the option of clicking on the run Query button, which will execute the query and return the selected data in the results grid. You will be able to see all of the fields and data associated with the query statement. Because different database have different field names for parameters, date/time information and results the user will have to align the database fields with the fields WASP needs information (i.e. Date/Time Field and Value Field). To align the data the user will specify which field in the database is associated with date/time by using the drop down pick list next to Date/time field, this pick list will contain entries for all fields in the user selected data file. The user will repeat this for the value field, which should be related to the numeric measurement or parameter value.

In some instances there may be duplicate data (multiple entries for a particular parameter, station and date) WASP has the option of averaging the multiple records (by checking the Average duplicate records option) or just using the last duplicate record.

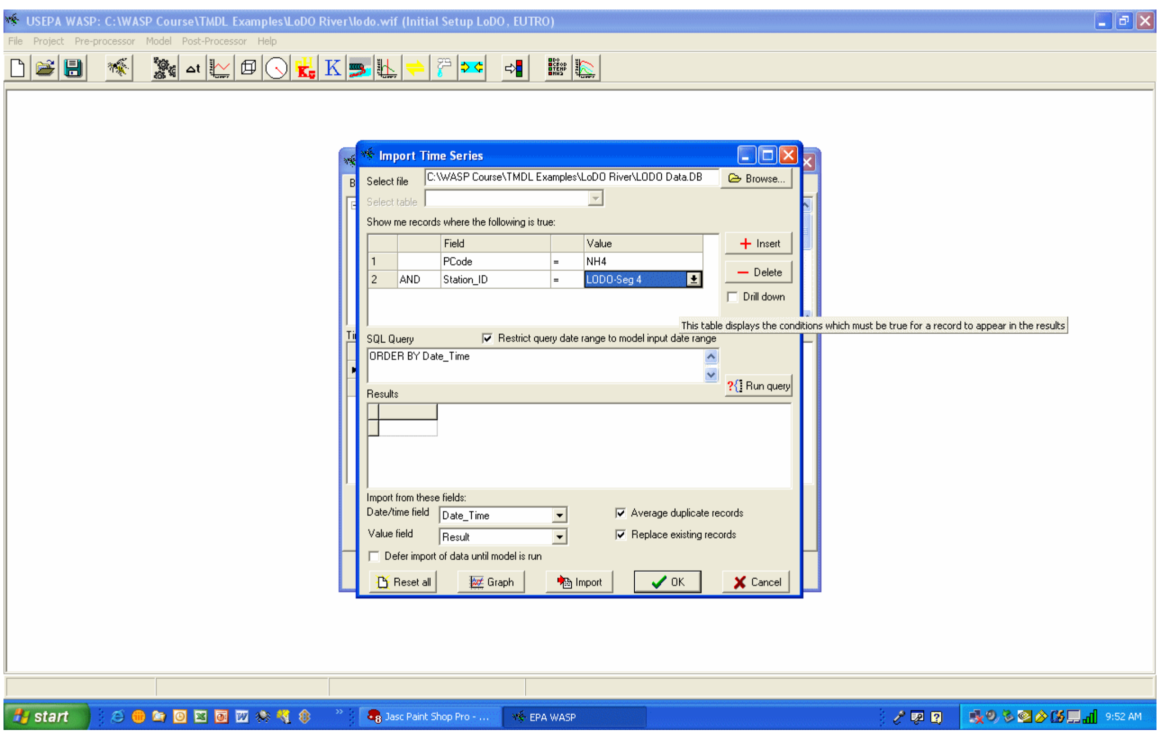


Figure 1-33 Database Query Grid for Time Series Extraction

Once the user has selected a file and developed the query in which to extract data, the preprocessor needs to know which fields are associated with date/time information and which field contains the numeric data to pull into the model. This is accomplished by selecting which field in the data file (via drop down list) that should be associated with the date/time values that will be imported (Figure 1-34). This field selection when using an Excel spreadsheet file as the source should correspond to the date/time column that is related to the results data (constituent data) that will be brought in for the particular time series being defined.

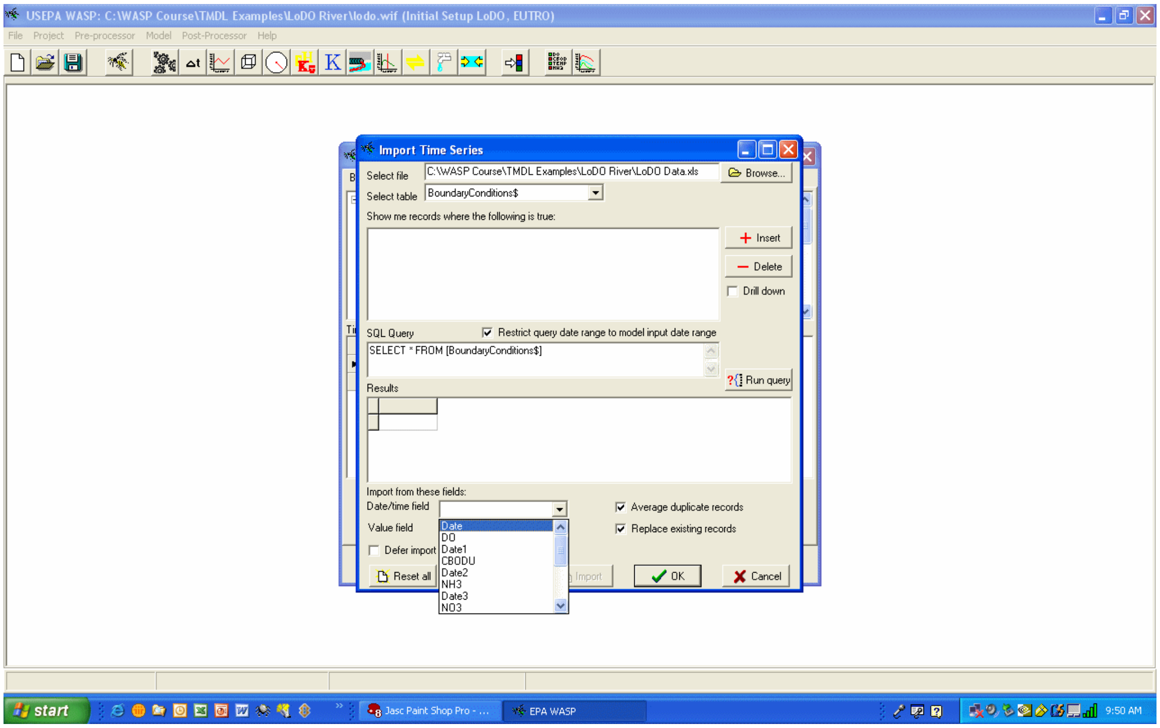


Figure 1-34 Time Field Selection

The same must be completed for the result field. In other words, the field that stores the numeric result that you want to use in WASP must be identified to the preprocessor. This is accomplished just like the date field where the user is presented a drop down list of available fields in which the numeric value for the Pcode of interest is stored.

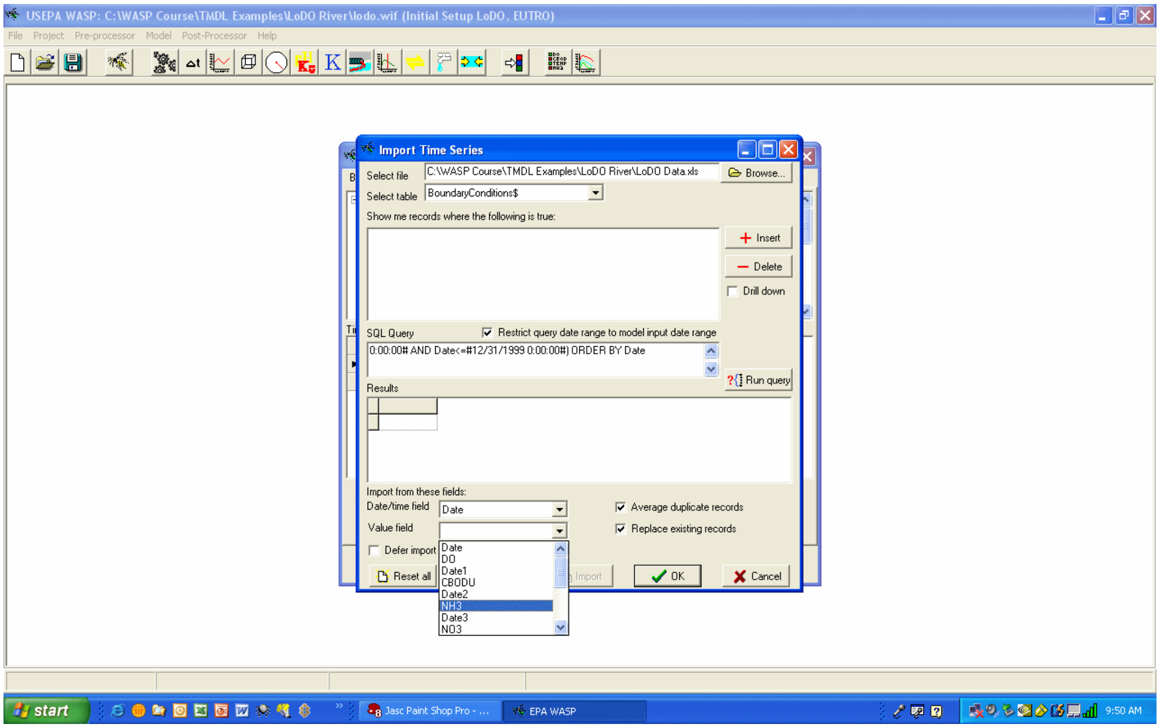


Figure 1-35 Result Field Selection

1.18.2. Defer Import

This is a very powerful option when bringing time series data into WASP. As the modeler begins to collect available data to be considered in the model application they can store it in a database or spreadsheet, the defer import option will allow the modeler to start adding data to the WASP knowing that additional data will be populated in the external data source. The defer option will allow this new data added to the external data source without the modeler doing anymore than adding it to the external files. This is of great use when initial input datasets are being developed as data is being assimilated and compiled into a central location. This allows initial model runs using the data as it is being assimilated and analyzed. As data is brought into the spreadsheet or database it will automatically be available to the model simulation.

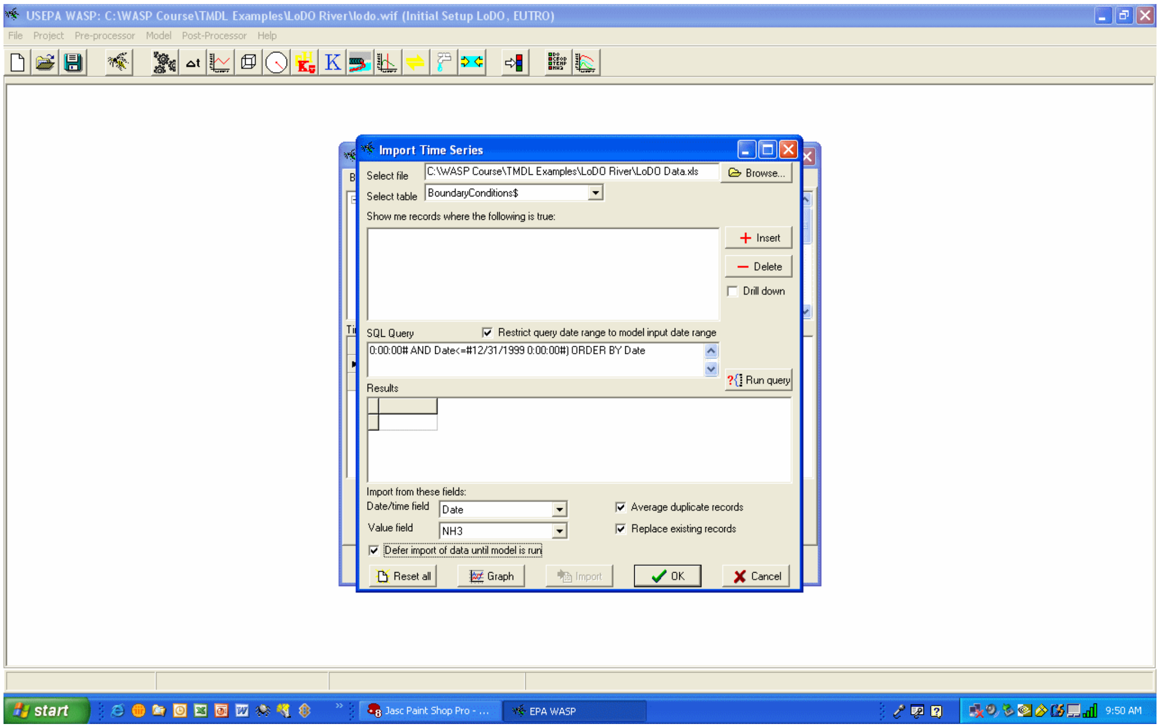


Figure 1-36 Defer Import Option for Time Series

If a deferred query has been defined for a particular time function in WASP, the user can determine this by the appearance of the data grid. If it has a cyan (blue) background this means there is query that has been deferred to run time for this time function. The user can click on the Import Button to see the query, edit the query, and import the query.

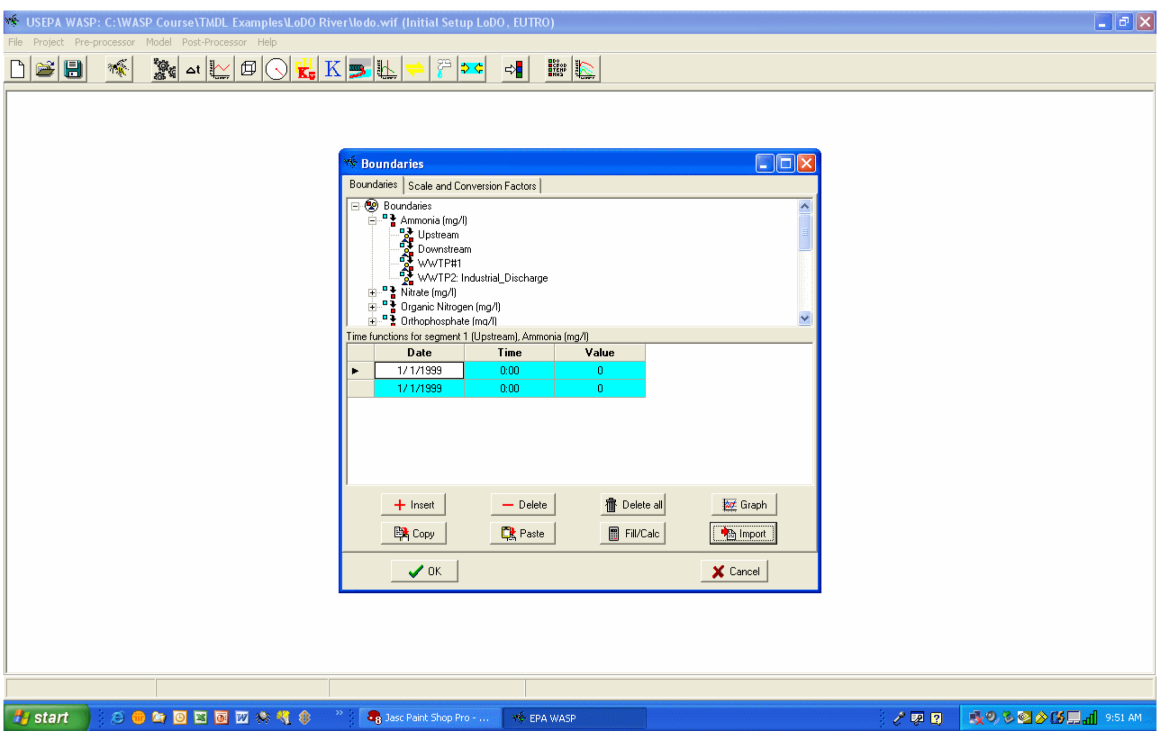


Figure 1-37 Defer Import Indicator

1.18.3. Import

Once the user has defined a query they have the option of executing the query and importing the data from the data source to the actual WASP input file. Figure 1-38 illustrates the dialog box that appears on the user selects import. This option should only be used when the data source that is being has been completely populated and verified.

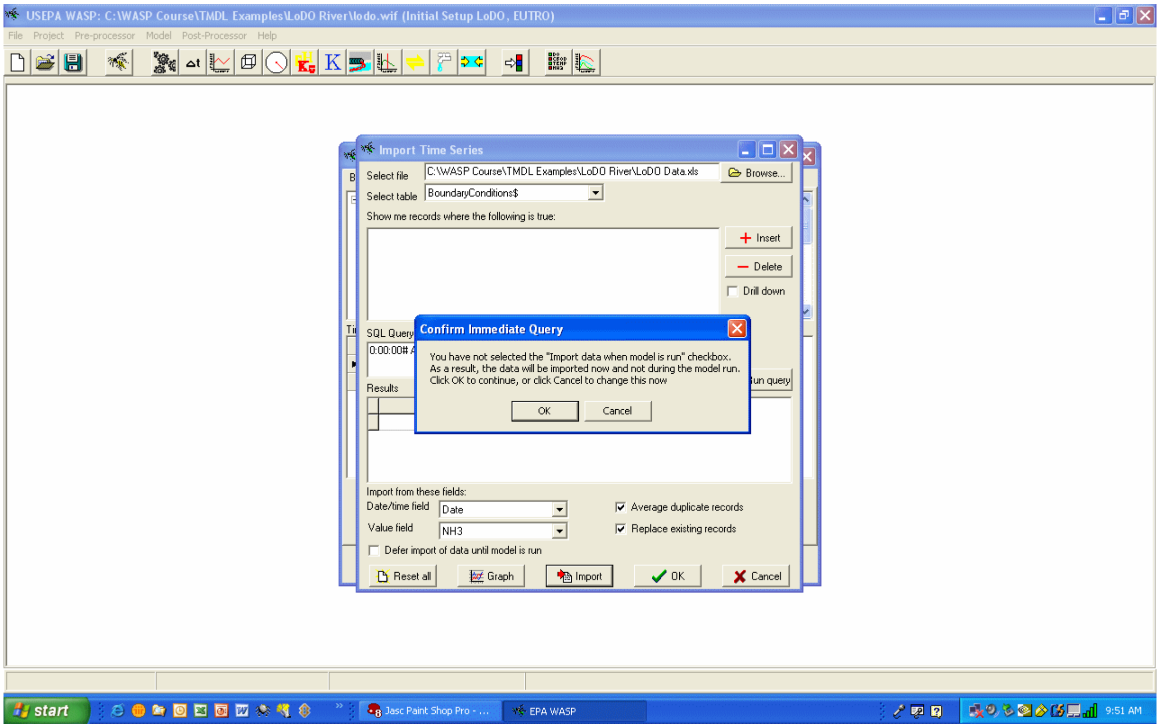


Figure 1-38 Time Series Import Function Dialog

1.19. WASP Network Tool

The WASP Network tool has several functions. The general purpose of the tool is to provide a quick method of getting external information into the WASP modeling framework. This tool can be used to get information from Geographic Information Systems (GIS), other models or tools. While the file formats for bringing information in through the Network tool is prescribed, it can be enhanced in the future to handle different file formats.

A standalone GIS tool has been developed that uses spatial coverages to help derive the WASP network (segmentation and connectivity). This tool uses several coverages to accomplish this task. The National Hydrology Datasets (NHD) is the primary coverage used. These coverages can be obtained from the USGS.

It is important to note before the user invokes this tool, the model type (EUTRO, TOXI, Mercury, etc.), and simulation start/end time should be set.

1.19.1. Control File

The control file contains the path and filename of each of the components of the WASP Network Tool. This file either needs to be created by the user or utility program that has been developed to take advantage of this function. Such an example is the GIS tool briefly described above.

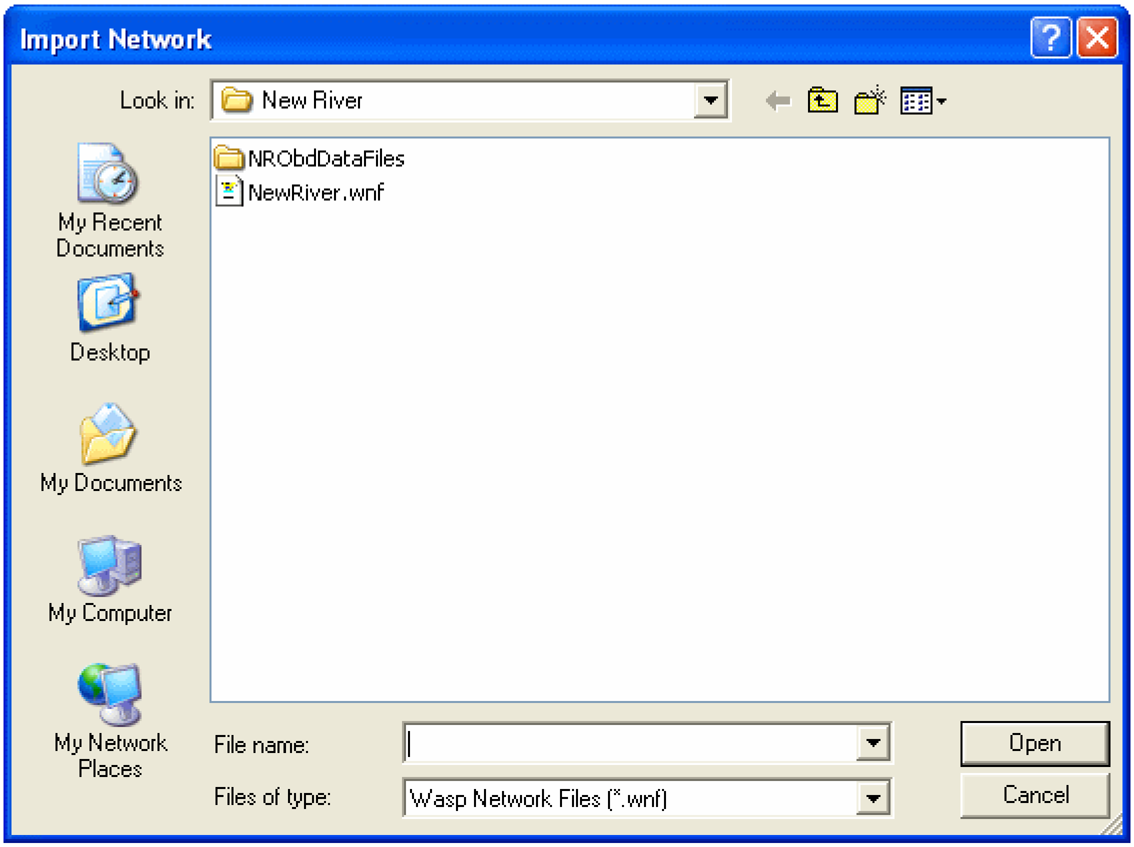


Figure 1-39 WASP Network Import Control File Selection

The control file should have the extension *.WNF (WASP Network File) for the file dialog box to easily find the file. The user can use different file extensions but would then be required to change the file filter to locate. This is a simple ASCII file that basically provides the path and filename to the three main file types that can be imported using this tool. An example of the file is given below:

C:\WASP Projects\GIS Data\Segment_Info.txt
C:\WASP Projects\GIS Data\Flow_Info.txt
C:\WASP Projects\GIS Data\Boundary_Info.txt
C:\WASP Projects\GIS Data\Load_Info.txt

Note the user is not required to use all files, if flows and boundaries are going to be imported the other lines should be left blank. If one of the files is not found the import function will be aborted.

1.19.2. Segment Information File

The segment information file contains information needed to define segment characteristics. The basic segment characteristics are: volume, depth, velocity, segment length, segment width, segment slope, and segment roughness. Any or all of this information can be contained in the import file. An example of this file is given in the examples directory in the Import File directory.

1.19.3. Flow File

The flow file contains information needed to the flow field functions in WASP. This file format pretty much matches the old DOS version input file format. Flow fields and individual low functions can be defined including the time series information. An example of this file is given in the examples directory in the Import File directory.

Note: For all time series information that is imported via these functions, time is Julian and must start with time equal 0.0, which corresponds to the start date and time specified in the model parameterization screen.

1.19.4. Load File

The load information file contains information needed to define loadings to individual segments for whatever state variable the user would like to use. This includes system and segment specification followed by the time series of loadings. Any or all of this information can be contained in the import file. An example of this file is given in the examples directory in the Import File directory.

1.19.5. Boundary File

The boundary information file contains time series information for boundary segments for each of the state variables. For the import function to operate successfully, the segment that is specified in the external file must match a boundary segment that is defined in the transport inputs (flows or dispersion). An example of this file is given in the examples directory in the Import File directory.

1.20. Output Control Database

Because the simulation result files that are processed by the graphical post processor (MOVEM) can get rather large depending upon model network and/or simulation time period, the user now has the option to specify which output variables get written in the simulation result file. By default all variables are selected. The user simple selects or de-selects the variables by clicking on the Output radio button located next to each output variable.

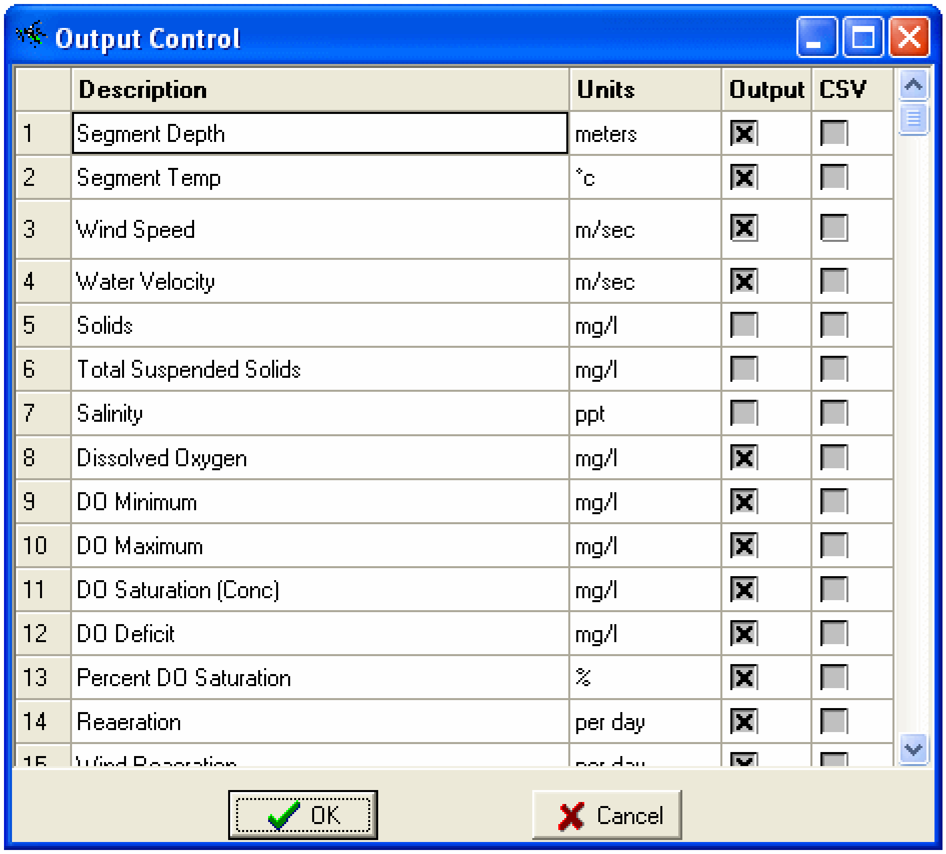


Figure 1-40 Output Control Screen

The user also has the capability of creating a comma separated value file of simulation results for any given output variable. A separate file is created using the output variable name for every selection. The CSV file will contain model results for each segment and output time. This CSV file can be directly opened by a spreadsheet file or a user developed utility program. The user should be aware that creating CSV files can slow the execution of the model substantially depending upon the size of the model network and simulation time period. The user should also be aware of the limitations in the size of files that can be processed by various spreadsheet programs.

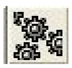
2. WASP Execution Supervisor

The purpose of the WASP Execution Supervisor is to allow the user to develop a control file that can be launched in a "Batch Mode". It allows the user to schedule a series of runs and have them occur without user intervention

2.1. *Toolbar*

The following icons are available:

 Open a previously created Control File

 Configure batch processing, add WIF files to run

 Save Control File

 Execute Batch Control File

The WASP Execution Supervisor (Figure 2-1) is relatively simple program that requires very little user interaction. To execute the WASP Execution Supervisor select it from the USEPA menu from the Start Menu.

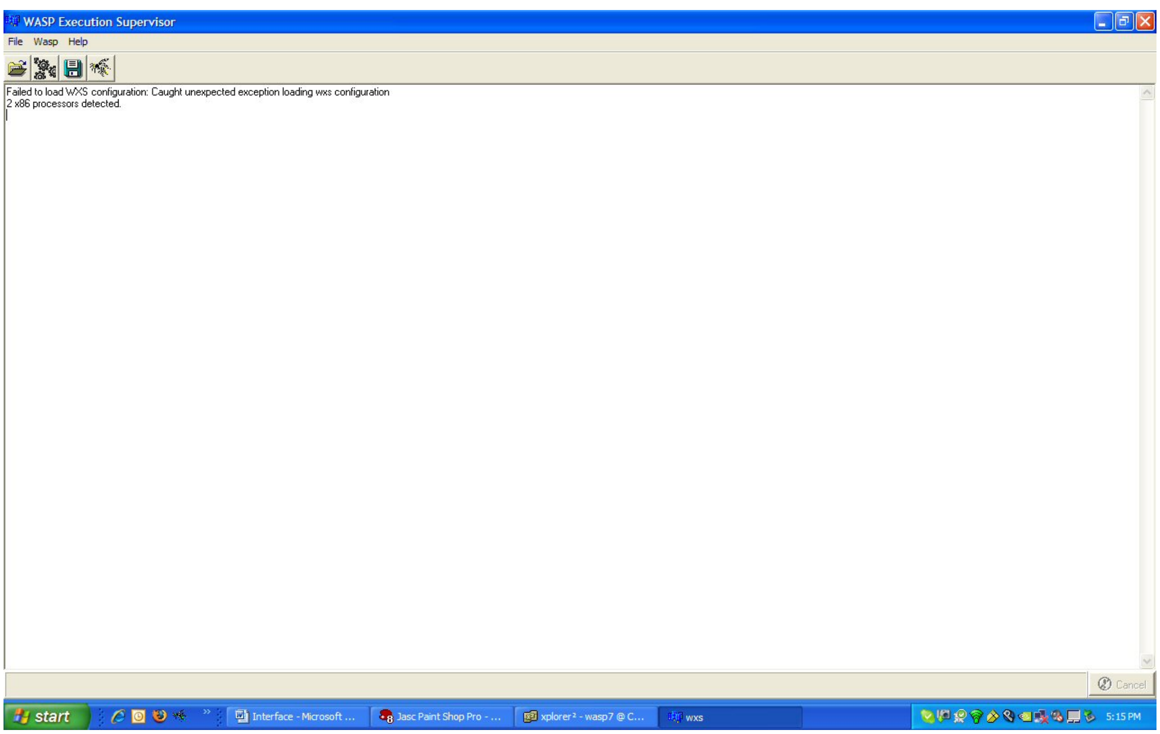


Figure 2-1 WASP Execution Supervisor Main Screen

2.2. Configuration

Configuration screen is where the user sets the run time parameters and determines which WASP Input Files (WIF's) will be included in the batch run (Figure 2-2). The user selects the configuration screen either from the menu or clicking on the icon.

2.2.1. Adding/Delete WIF Files

To add WIF files to the configuration screen to build a batch file of runs, click on the insert button and browse to where the WIF files are located. Select the WIF file and click on okay and it will be added to the list.

> Note: You can add more than one file at a time by holding down the "Shift Key" (to select a range of files) or the "Control Key" (to select individual files), once highlighted click Okay and they will appear in the queue.

To delete a WIF from the batch list, select the row of the file you wish to delete and click on the Delete icon.

2.2.2. Setting the Number of Simultaneous Simulation

The user can set the maximum number of WASP runs occurring simultaneously. This number will vary from computer to computer. If you machine is a dual processor or dual core cpu, you will find that 4 simultaneous executes runs faster than 2.

> Note: Even if you are using a hydrodynamic linkage file or queries from a database you can have more than 1 WASP run accessing this data.

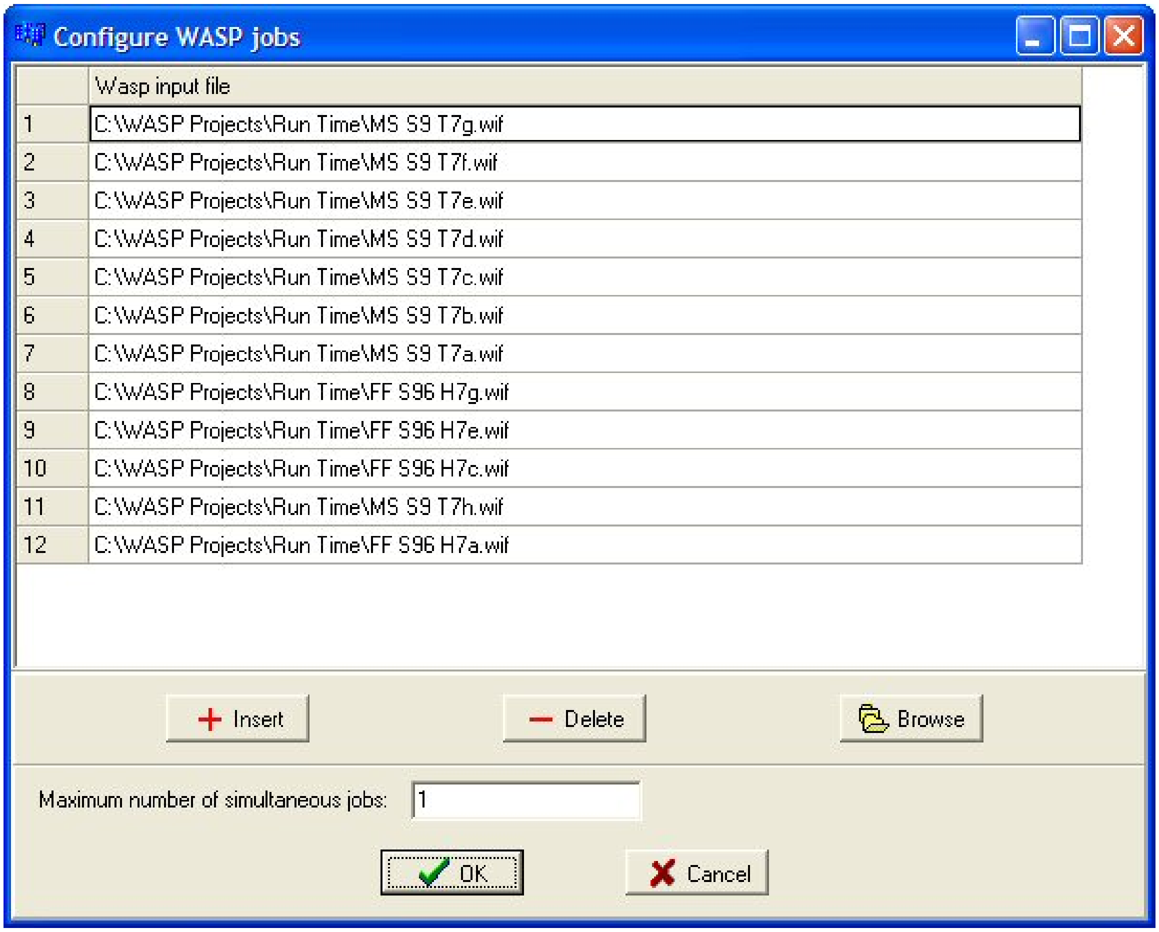


Figure 2-2 WASP Execution Supervisor Configuration Screen

2.3. Save Configuration File

Once completed with the configuration, the user can save the file as illustrated in Figure 2-3. These files can be stored anywhere on your system, the full pathnames are stored for the WIF files.

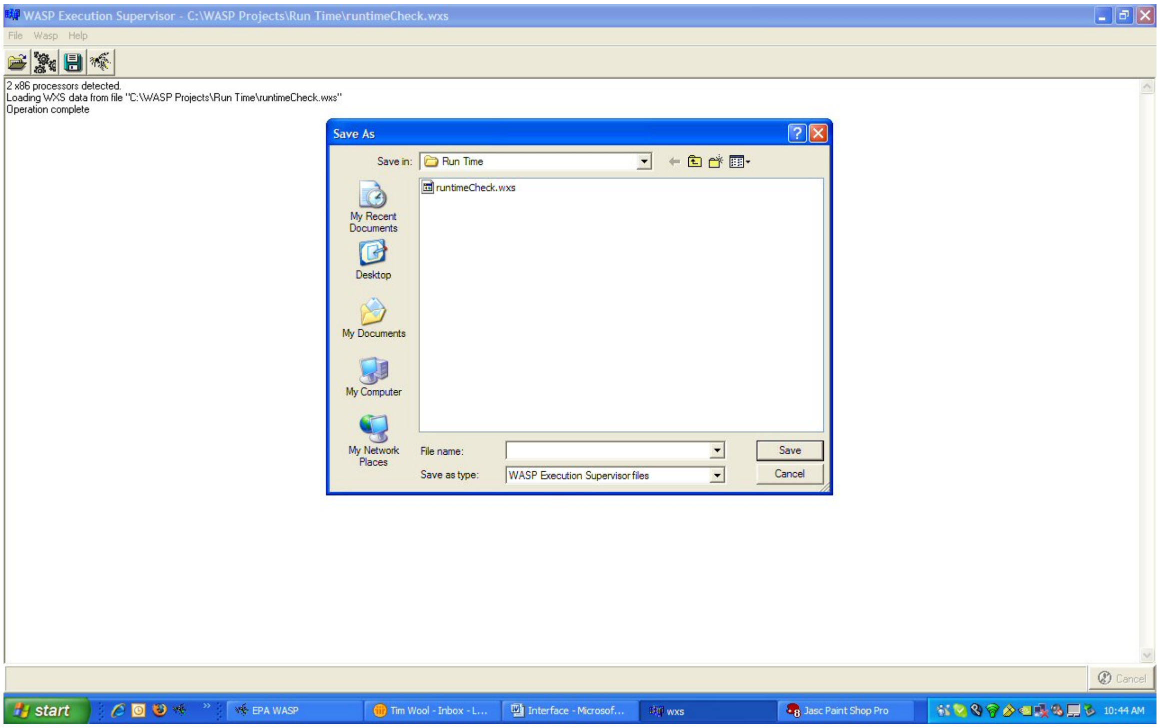


Figure 2-3 WASP Execution Supervisor Save Configuration

2.4. Execution

With a properly configured configuration file you can start the execution sequence from the menu system clicking on WASP or you can click on the run icon. Once the run is selected you will be presented with a screen as illustrated in Figure 2-4. the model threads will be created for the maximum number of simultaneous runs, you will see the WASP splash screen pop up, the model will run in the minimized mode not to disturb your other activities. Once each of the individual model simulations has been completed a message is returned to the run time screen.

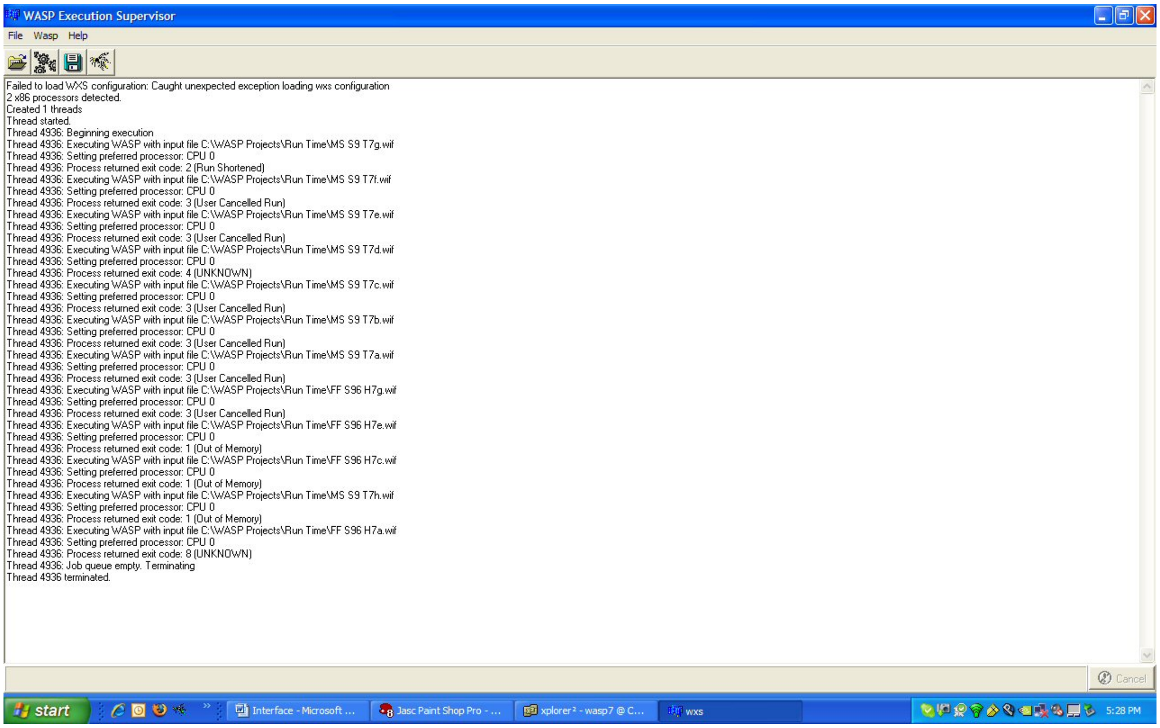


Figure 2-4 WASP Execution Supervisor Execution Screen

2.4.1. Error Codes

If an error occurred during execution it will be returned to the runtime screen. The errors that are currently being tapped are:

1) Run Successful – No problems
2) Out of Memory – Reduce number of simultaneous simulations
3) Run Shortened – WASP Aborted, run WIF file standalone to determine error
4) User Cancelled Run – You hit the abort button

3. Visual Graphic Post-Processor

The Post-Processor was developed as an efficient means of processing the vast amount of data produced by the execution of the WASP models. It has the ability to display results from all the models (EUTRO and TOXI) included in the WASP modeling package. The Post-Processor reads the output files created by the models and displays the results in two graphical formats:

1) Spatial Grid – a two dimensional rendition of the model network is displayed in a window where the model network is color shaded based upon the predicted concentration.

2) x/y Plots -- generates an x/y line plot of predicted and/or observed model results in a window.

There is no limit on the number of x/y plots, spatial grids or even model result files the user can utilize in a session. Separate windows are created for each spatial grid or x/y plot created by the user.

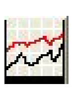
The Graphical Post-Processor is routinely executed from WASP. Also, the user can use the Windows Explorer or Run button to execute the program. If executed from within WASP + with an input file selected, the corresponding model output files will be loaded. If executed from within WASP without an input file selected or by some other means, the user will need to use the file options for opening the files they want to display.

3.1. Main Toolbar

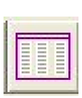
There are several toolbars and speed menus available. The main tool bar is available below the pull down menus provide the following functionality to the user. Depending upon the current status, some icons may not be available to perform a task, thus are not active.

Open File Icon. This initiates the open file dialog box that allows the user to open a model result file (*.BMD), geometry backdrop file (*.BMG) or observed data database (*.DB).

Creates a Spatial Animation Window using GIS coverage's. This option is only available when GIS coverage's have been opened. One of the GIS coverage's required is model network coverage.



Creates x/y plot Window. This opens an x/y plot window only after model data (*.BMD) or observed database data (*.DB) have been loaded. The user can open as many of these windows as desired to review any data that is loaded.



Edits the load observed data database

3.2. *Model Output Selection*

The Graphical Post-Processor was designed to allow the user to rapidly evaluate the results of the WASP model simulations and its support programs. Observed data can also be stored in a database format.

Three types of data are recognized:

> The first data type is created from the execution of the WASP models (*.BMD). The output from WASP is written in a binary file format. The model results cannot be read directly by any other program.
>
> The second file type that can be read is a Paradox table file (*.DB). The Paradox table file is used to provide observed/field data to be plotted against model predictions.
>
> The third file type is an ArcView shape file. These files can be used in the spatial analysis mode to aid the user in displaying the model network with respect to its geography and surrounding characteristics.

3.2.1. Opening Model Output

Prior to working with any model data or observed data, the files must be selected by the user. There is no limit to the number of files that can be opened. If the user would like to open additional files, the procedure given below will illustrate how to load each of the different file types.

To open a file, the user can use the menu system and select open file or press the open file icon. This will display a file dialog box as illustrated in Figure 3-1. From this file dialog box the user can navigate to any drive and directory to which their computer is attached. By pressing the down arrow on the file type dialog, a list of valid file extensions is displayed for the user. Selecting an extension will result in the display of a picklist of the available files in the current drive and directory.

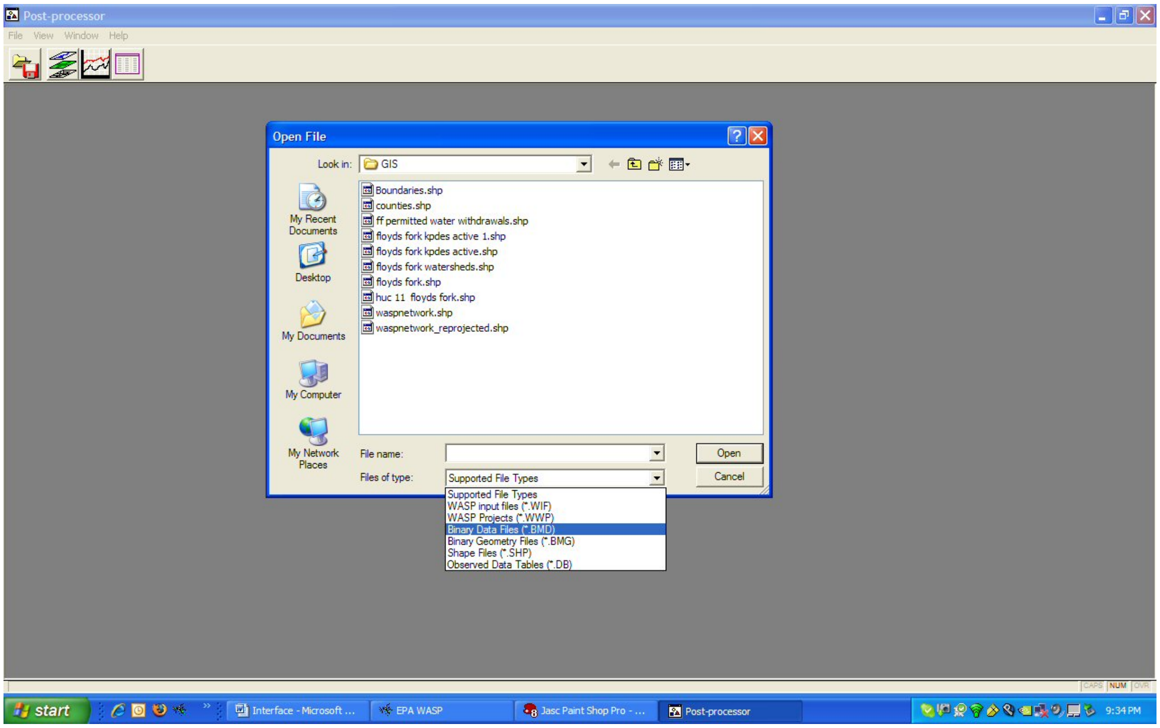


Figure 3-1 File Dialog Box

BMD Format

To open a WASP simulation result file, select binary model data from the file type box. This will cause the file dialog box to display only those files that have the extension *.BMD. The user has the ability to move around between drives and directories to select a file to review. The user can either double click the mouse on the desired file or highlight the file and press the open button. Once the file is open the x/y plot icon will become available.

> Note: The user must load a binary model geometry file (*.BMG) before the spatial grid analysis icon is available.

Observed Data

The observed data is in a Paradox 4.5 or higher format (*.DB). In order to display observed data versus the predicted model results, the database must be in a specific form. To load an observed data database follow the procedures described above, change the file type to *.DB. Select the database and press open.

> Note: The "observed data" database is expected to be in a certain file format with pre-defined field names.

3.3. Spatial Graphical Analysis

The spatial graphical analysis allows the user to review model results for the whole network for a given constituent and time. This mode of graphical representation of the model results is very effective in illustrating model predictions to non-technical audiences. This option requires the user to have a ArcGIS shapefile of their model network that is appropriately formatted for MOVEM.

3.3.1. Overview

The spatial graphical analysis function allows the user to illustrate the model results on a spatial grid using shading to represent predicted values. The spatial grid analysis provides three modes for looking at the model results: shaded, wired frame, and violation/criteria shading. These various modes allow the spatial graphical analysis mode to illustrate information from model simulations in such a manner to make easier for the non-technical person to understand the results. The following section provides information on how to use and configure the spatial graphical option.

To initiate a spatial grid, the user should press the spatial grid analysis icon: this will generate a spatial grid window as illustrated in Figure 3-2. The user can create as many of these windows as desired. Each window can be configured to show different model results as well as different modes.

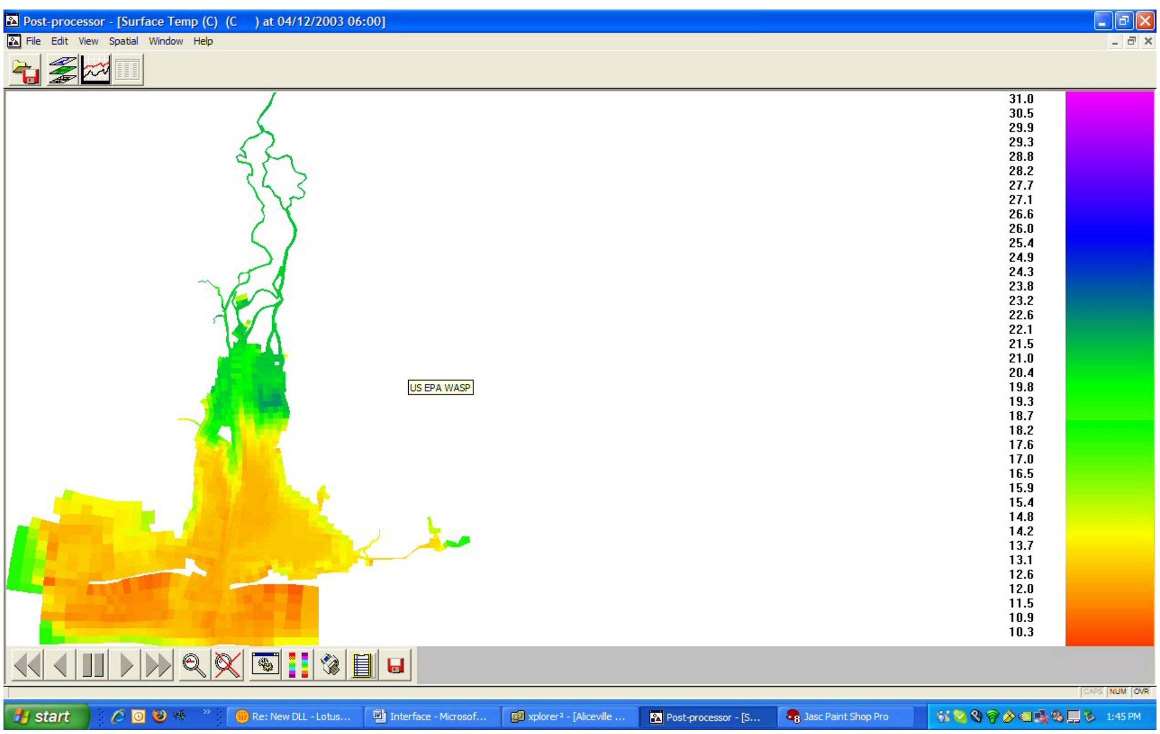
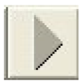


Figure 3-2 Spatial Analysis View

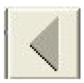
3.3.2. Spatial Grid Toolbar

The second option for the spatial analysis tool is to allow the user to develop the model network using ArcView and combine the model network with other GIS coverage's. To use this method the user must have a copy of ArcView and good working knowledge of how to use it.

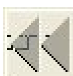
Each spatial grid that is generated by the user has its own set of controls that allow the user to manipulate the contents. A description for each of the icons on the spatial grid toolbar are given below:



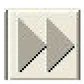
Step Animation Grid Forward One Time Interval. This button causes the active spatial grid to step forward one time interval in the model result file.



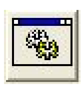
Step Animation Grid Back One Time Interval. This button causes the active spatial grid to step backward one time interval in the model result file.



Plays animation sequence forward in time.

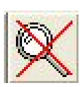


Plays animation sequence backward in time.



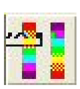
Configure Animation Grid. This button allows the user to access the spatial grid configuration screen where the user can select what backdrop file and model data will be displayed.

Un-zooms the spatial plot one zoom level at a time



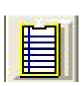
Un-Zooms the spatial plot to the full extent

Configuration dialog box for spatial plot

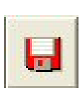


Invert/Revert the color scale of the legend

Print the current spatial plot to a printer



Copy the current spatial plot to the clipboard



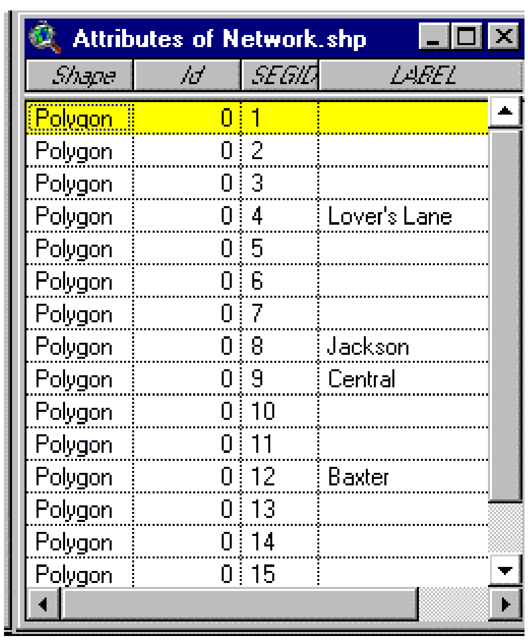
Save the current spatial plot as a Windows bitmap (BMP)

3.3.3. Geographical Information System Interface

The graphical post processor has the ability to display ArcView shape files with the model network being one of the layers. To use this option the user must have access to GIS coverage's and a copy of ArcView. The post processor can display any number of coverage's, the shape files must be loaded using the file open option or be contained in the project file. Before a model network can be displayed the user must develop a coverage and related database using the ArcView program.

Creating Coverages

To create a coverage the user will need to rely on a good understanding of ESRI's ArcView/ArcInfo programs. The model network will need to be added as a layer (Theme) in ArcView. The best method for creating this layer is to create a base map layer out of ArcView. This layer should contain the waterbody being modeled my WASP. Draw the segmentation on the printed map. If the segmentation includes sub-surface segments, you may need to create yet another layer. Once the segmentation has been drawn on the base map, place the map in your digitizer and register the map following the directions provided in the online help. Once the map has been registered, the user must digitize the segment polygons. To do this the user must add a new theme, with a separate polygon for each segment. The user will need to create a database table of attributes for this layer. This database contains basic information that the post processor uses to align the model predicted results with the correct segment. When creating this table the user is required to have at least one field, SEGID. This field must be called SEGID and be a string field. An optional field that can be added is LABEL. LABEL is a string field as well. The LABEL field can be used to put an alphanumeric description next to the segment on the spatial plot. The user is referred to the ArcView documentation on how to create, edit and modify tables.



Shape	Id	SEGID	LABEL
Polygon	0	1	
Polygon	0	2	
Polygon	0	3	
Polygon	0	4	Lover's Lane
Polygon	0	5	
Polygon	0	6	
Polygon	0	7	
Polygon	0	8	Jackson
Polygon	0	9	Central
Polygon	0	10	
Polygon	0	11	
Polygon	0	12	Baxter
Polygon	0	13	
Polygon	0	14	
Polygon	0	15	

Figure 3-3 Model Database Definitions

3.3.4. Controlling Spatial Analysis

Each of the spatial grid analysis windows can be configured several different ways to display the model results. The user can elect to display results in one of three animation modes:

1) Time – this mode animates the results for a single variable forward/backward in time. This is the preferred mode of the spatial analysis grid.

2) Variable – this displays the shaded results for each of the variables in the model output file. Pressing the forward/backward icon will cause the spatial animation grid to display the results for each of the variables.

3) Slice – this option displays the results for a given variable over the range of slices views found in the BMG file.

Figure 3-4 illustrates the configuration screen for the spatial analysis grid non-GIS.

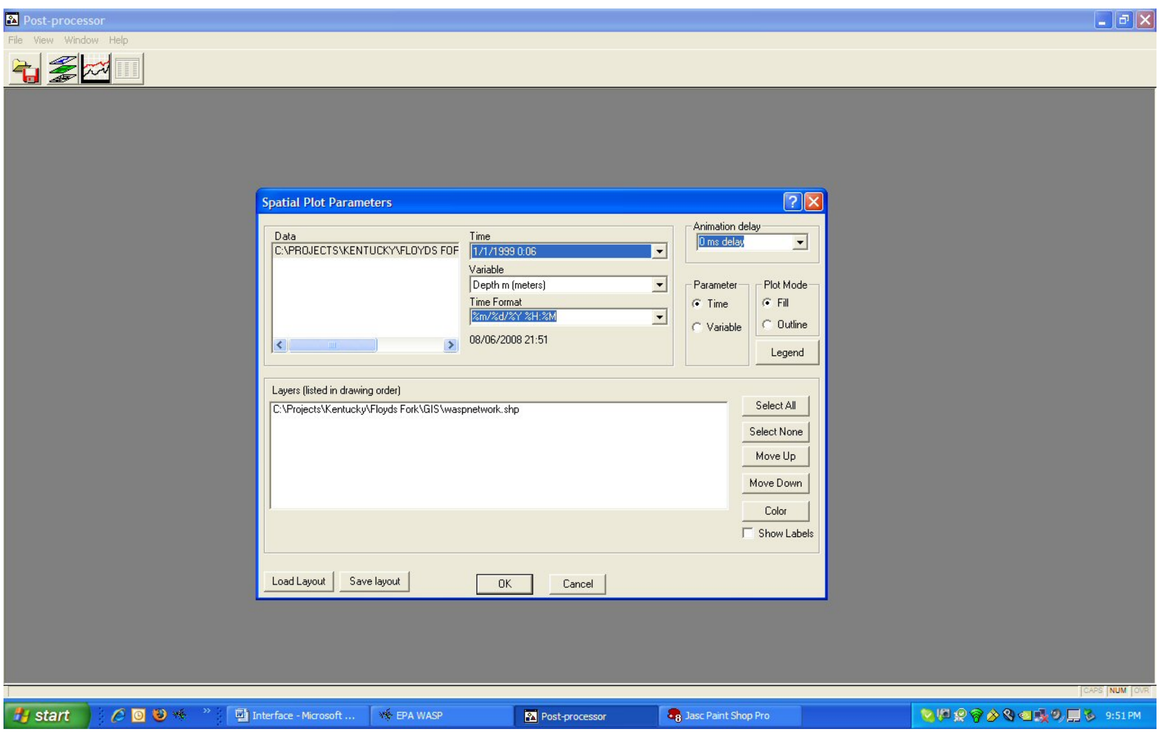


Figure 3-4 Spatial Analysis Configuration

The GIS option works much like the other, with the exception of the ability to display individual GIS layers.

3.3.5. Selecting Dataset

Because more than one model simulation result file can be loaded at a time, the user has the ability to select which of the model results files to display. Each spatial analysis grid can display only one model result file.

To select a currently loaded dataset to display in the current spatial analysis grid, press the Configuration Button and select the drop down picklist for Data Set. This will make the results from the selected file appear in the spatial grid.

> Note: The spatial grid analysis can only display information from one model result file at a time. The user has the option of creating as many spatial analysis grid windows as needed.

3.3.6. Selecting Variable

The user has the ability to select any one of the simulation result variables that is written to the output file by the execution of the WASP models. The model result file (EUTRO or TOXI) will determine the variables that are available for display. To select a variable for a selected output file use the variable drop down picklist and select the variable you want the spatial analysis grid to display.

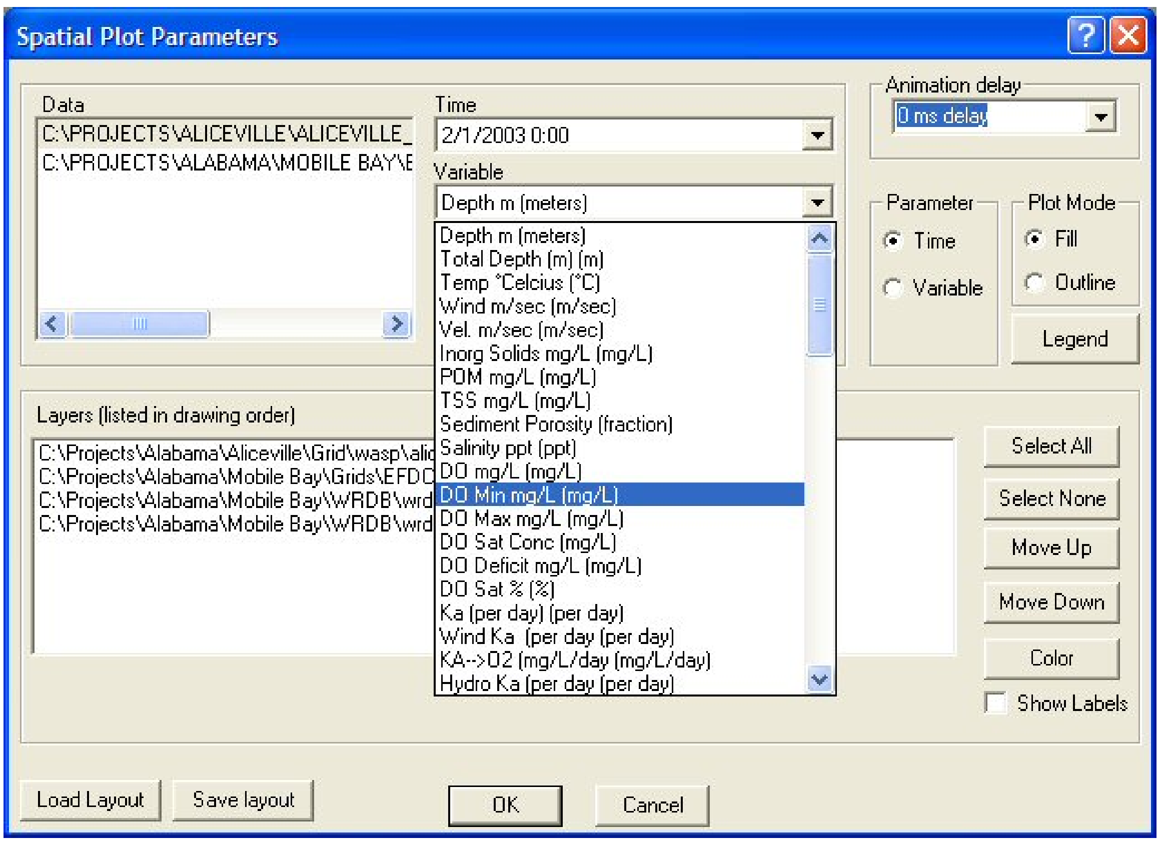


Figure 3-5 Selecting Variable to Display

> Note: The spatial analysis grid can only display one result variable at a time.

3.3.7. Selecting Time

The spatial analysis grid displays results for a given variable at a given time. Not only does the user have the ability to control the display of variables, but can control the display time as well. The user can select what simulation time to display the results for a given variable. This pull down picklist allows the user to move the spatial analysis grid to specific points in time without stepping through each time interval.

3.3.8. Palette

The user has the ability to modify the color palette that is used to shade the plot. The most common modification to the color palette would be to invert the currently loaded palette. Clicking on the Invert radio button will cause the color palette to be inverted (i.e. the colors for the lower values will become the colors for the higher values).

The user also has the ability to select a different color scheme by selecting a new palette. To select a new palette the user should press the browse button and select a palette.

3.3.9. Animation

The most common use for the spatial analysis grid is to animate the model predictions. When the forward/backward buttons on the toolbar are pressed, the spatial analysis grid is updated with the next time or variable depending upon the varying parameter. By continually pressing the forward button the user can create "a movie" of the model-predicted results. Alternatively, the user may press the movie icon. The speed of the animation is controlled by the options under the spatial plot parameters.

3.3.10. Plot Mode

Three modes are provided by which data can be displayed in the spatial grid analysis window:

Shaded

This mode displays the simulation results by shading the model computational element based upon the predicted concentration. A color legend is displayed on the right hand side of the spatial analysis window, and the computational elements are shaded in accordance to the legend based upon the concentrations predicted by the model.

Wire Frame

The mode displays the simulation results in a wire frame. This mode differs from the shade mode in that the model computation elements are not filled with a color based upon predicted concentrations. The wire frame mode outlines the outside perimeter of the computational element in a color corresponding to the predicted concentration.

3.3.11. GIS Configuration

If the user creates a GIS spatial view the controls and options for configuring the windows are little different. Figure 3-6 illustrates the configuration screen for the GIS option.

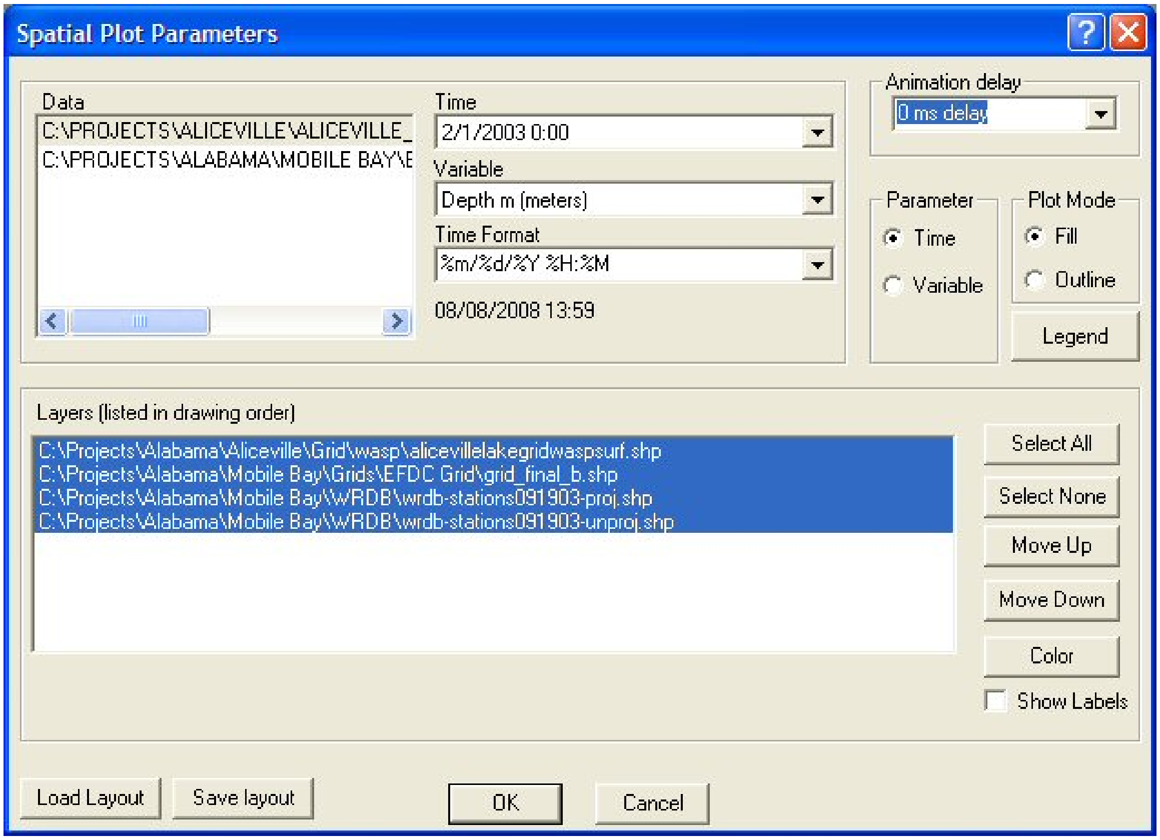


Figure 3-6 GIS Spatial Plot Configuration

> Note: GIS layer files must be loaded prior to this option being available.

3.3.12. Layers

The user can select and order that GIS layers will be displayed. The user is given a dialog box that contains a list of all the loaded layers; the user can select which one to display by using standards Windows selection methods. Once a layer has been added to the selected dialog window, the user can control the order in which the layer is drawn by moving the layer up and down in the window. The model network should be the last layers to be drawn to make sure that another layer does not over write it.

Select All

This will select all of the loaded layers. Once they are selected the user should press the Add Button.

Select None

De-selects any selected layers.

Move Up/Down

The move/up down button will move the selected (highlighted) layer up in down in the list. The layers at the top of the list are drawn first.

Layer Color

The user has the ability select a specific color for a given layer. Because the GIS layers do not have a color associated, the post processor assigns a color. This assignment may not be desirable, to change the layer color select the layer and press the layer color button. The user will be presented a color selection dialog box. Select the color and press Okay.

Legend

The legend by default is automatically configured based upon the range of data being displayed. The user can elect to set the range of colors used for a particular value in the legend definition screen. Manipulating the color scheme of the legend can aid in displaying the impacts of management scenarios.

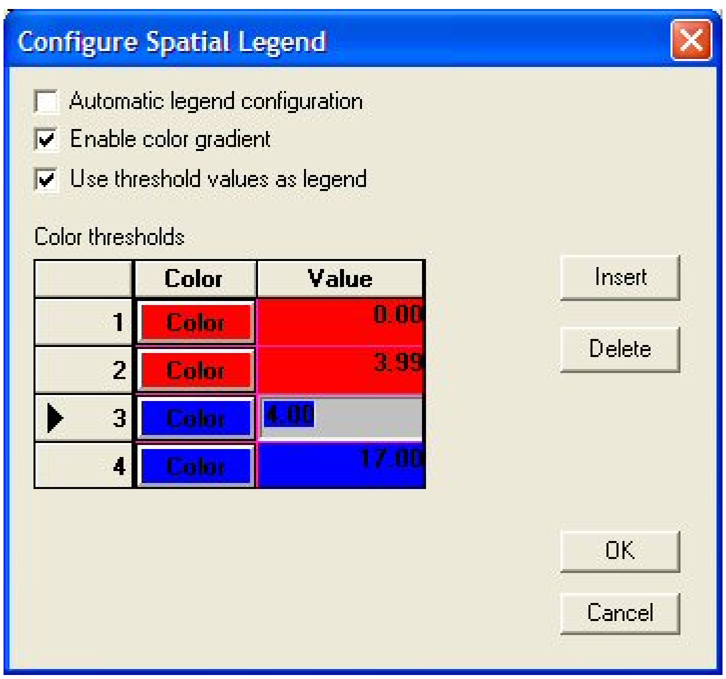


Figure 3-7 MOVEM Spatial Plot Legend Configuration

> Configuring the legend to match water quality standards or cleanup levels is a great way to show decision makers the management scenarios

Save/Load Layout File

Once you have a spatial plot window configured in a particular way, you have the option of saving the layout to retrieve later using the Load Layout button. The layout includes content: variables by position in the file, zoom level, labels, selected GIS layers and legend settings.

3.4. x/y Plots

The x/y plot mode represents the conventional method by which scientific data is displayed. While x/y plots are the conventional mode, the flexibility and control the user has over the way the x/y plots are configured is not. The user is provided as much flexibility as possible when developing x/y plots. The user can plot different model results files simultaneously, multiple variables and observed versus model predictions.

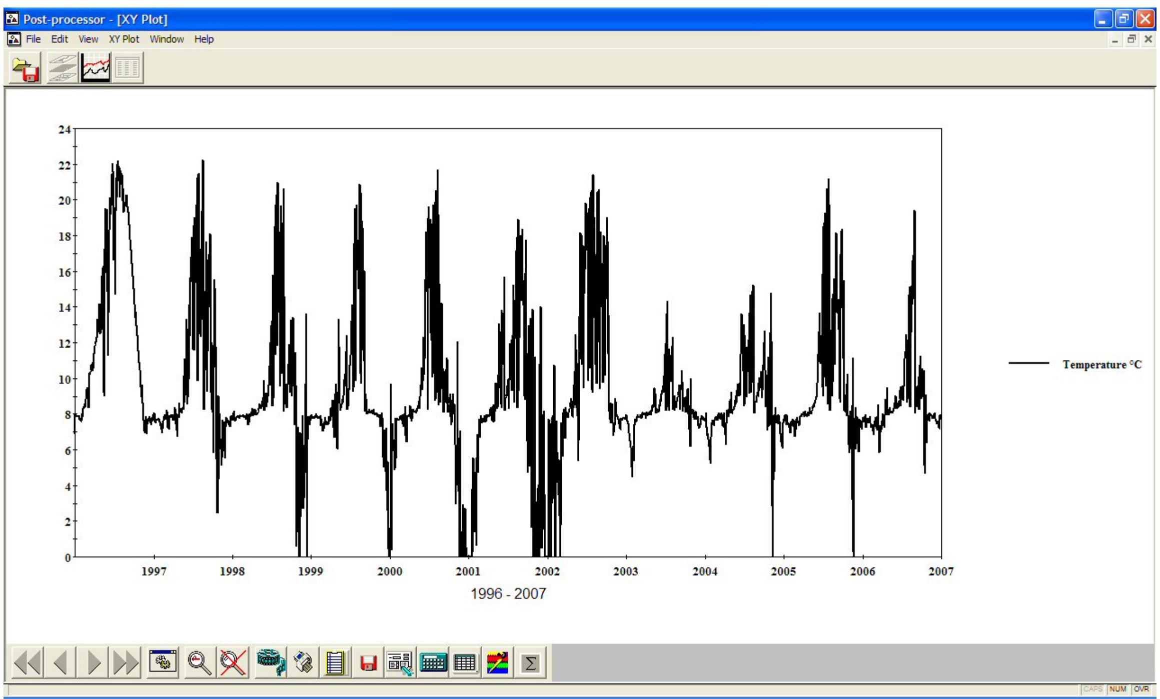
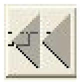


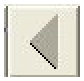
Figure 3-8 Example Graph

3.4.1. Toolbar

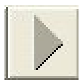
The x/y plot window has its own set of controls that allows the user to perform the various options that are available. The user can access these options either via the tool Pan x/y plot all the way to the left bar or the speed menu. The following is a description of the x/y plot toolbar:



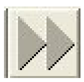
Pan x/y plot all the way to the left



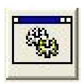
Pan x/y plot left



Pan x/y plot right

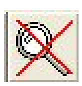


Pan x/y plot all the way to the right



Configure x/y Plot. This icon is used to configure what data is used and how it is displayed in the x/y plot.

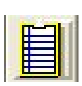
Un-Zoom One Level. This option will un-zoom the graph one user performed zoom level at a time. It undoes a zoom one step at a time.



Un-Zoom x/y Plot. This icon will un-zoom the graph to its original axes dimension at the time of creation of the graph.

Animate x/y Plot. This option is available when there is more than one line on the graph. It will toggle the display one line on the graph at time for visual inspection. It is used to "declutter" the graph to see differences.

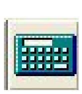
Print x/y Plot to Printer. This option allows the user to print the x/y plot to a printer. A normal Windows printer dialog box will appear to allow the user to control the appearance of the graph.



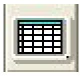
Copy Graphic Image to Clipboard. This option makes a copy of the x/y plot onto the Windows clipboard. This clipboard image can then be pasted into programs like a word processor for publication.

Save Graph to Disk. This option creates a Windows bitmap file (*.BMP) of the x/y graph window. The saved BMP can be imported into programs like a word processor for publication.

Export Graph to ASCII Table. This option creates an ASCII file containing the values for each of the lines from the graph window. This table can be imported into a spreadsheet or other programs.

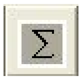


Curve Calculations. This icon brings up the curve calculation dialog box. The user can perform data transformation on the data or use one of the predefined functions.



Create Data Table from Graph. This creates a table of data represented by the lines in the graph window. The user will be able to review the data and copy data from the table to programs like spreadsheets.

Toggle graph window between color and black & white. Because some printers do not transpose the color to black and white effectively, this option allows the user to force black & white or color.



Calculates calibration statistic when there is one model prediction curve and one observed data curve

3.4.2. Creating x/y Plot

The first time that the x/y plot icon button requesting a x/y plot is pressed, the graph configuration menu appears. It is from this menu that the user has control over the content of the x/y plot window. The user can plot data within this x/y window for any or all of the currently loaded files (simulation result files or observed data). The user has full control over the appearance of graph (axis, title and legend labeling).

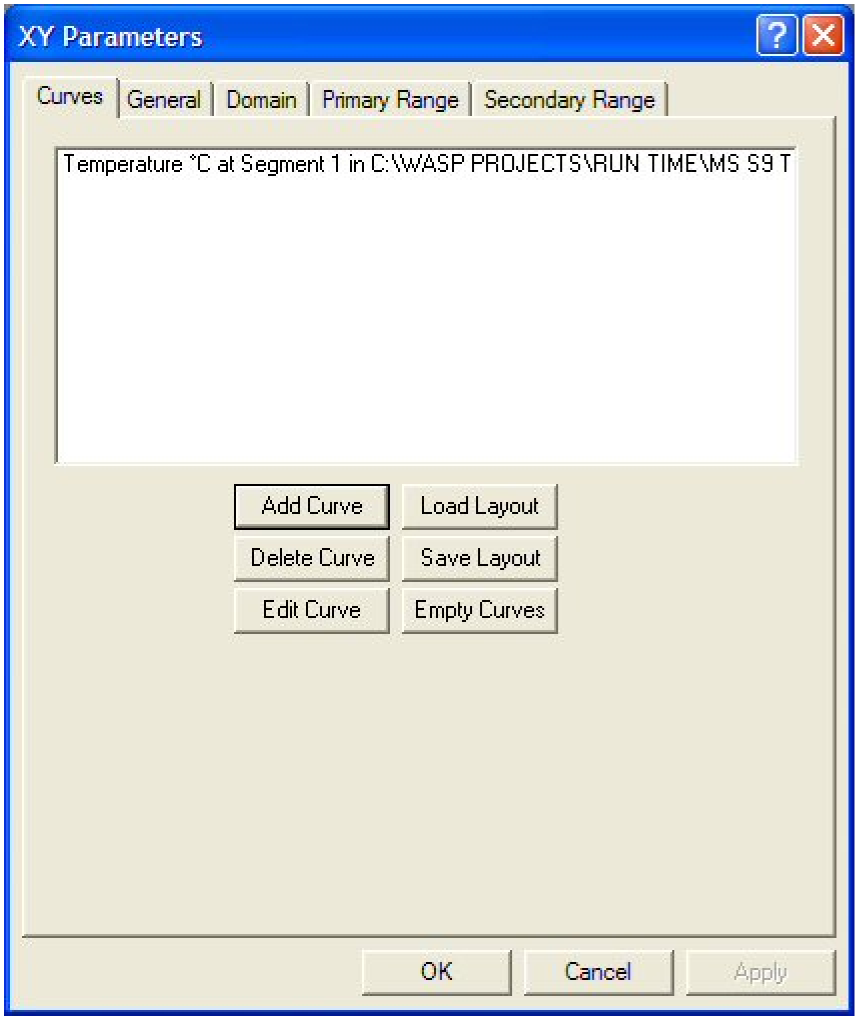


Figure 3-9 Graph Curve Attribute Screen

Graph Characteristics

The user has the ability to define the style of the x/y plot window. The user can define line types for the grid, colors for the various portion of the graph and control the fonts for the various text components. Once the user develops a style they have the ability to save this style to disk. Each individual x/y plot window can recall the style. The user can define a default style that is automatically recalled every time an x/y plot window is created.

The user specifies the title for the graph in this dialog box and can control the animation speed.

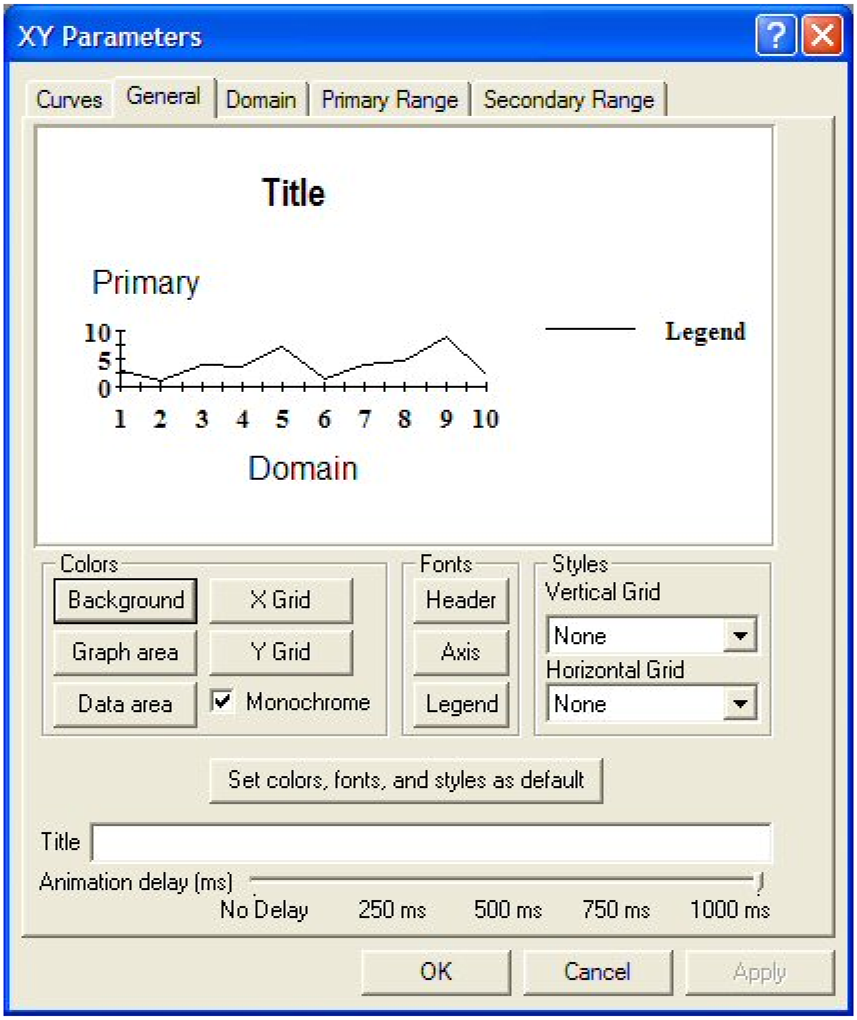


Figure 3-10 x/y Plot Characteristics

Domain Label

This dialog box is used to describe the label that will be displayed below the x-axis on the graph. Typically, the x-axis is used for either time or distance.

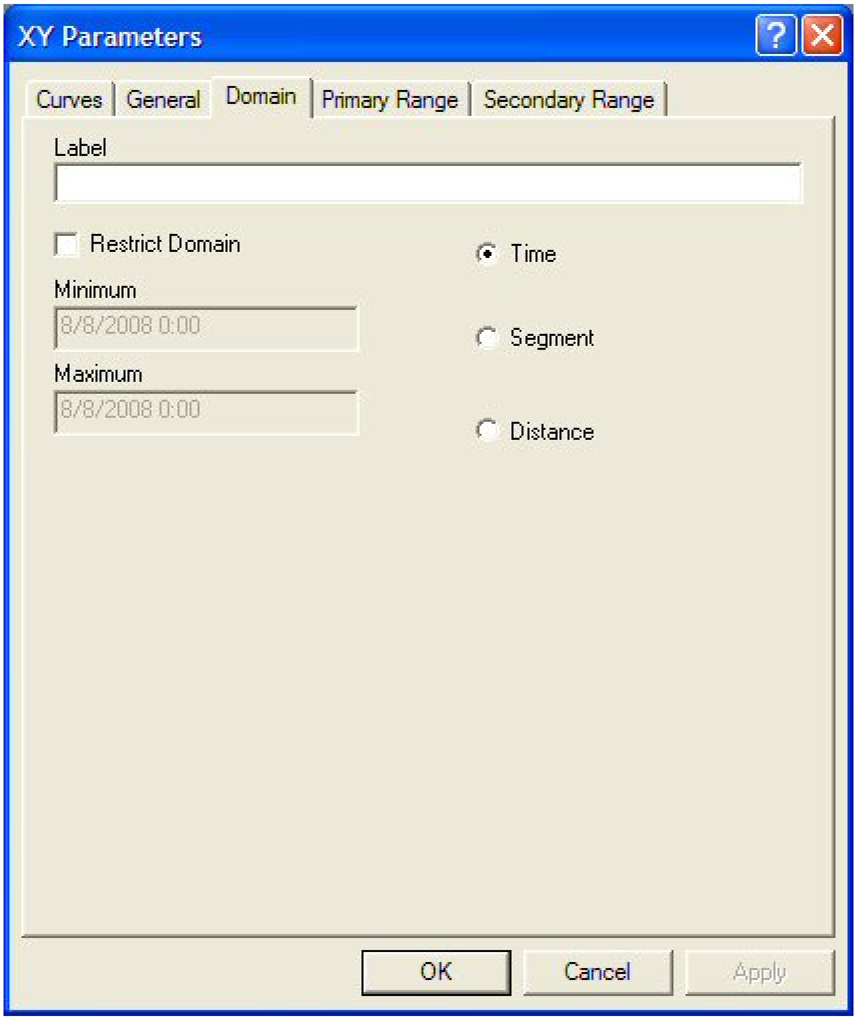


Figure 3-11 Graph Domain Labeling

Primary Range Label

This dialog box is used to describe the label that will be displayed above the y^1 axis on the graph. The y^1 axis is the one on the left-hand side of the graph and is typically used for concentration.

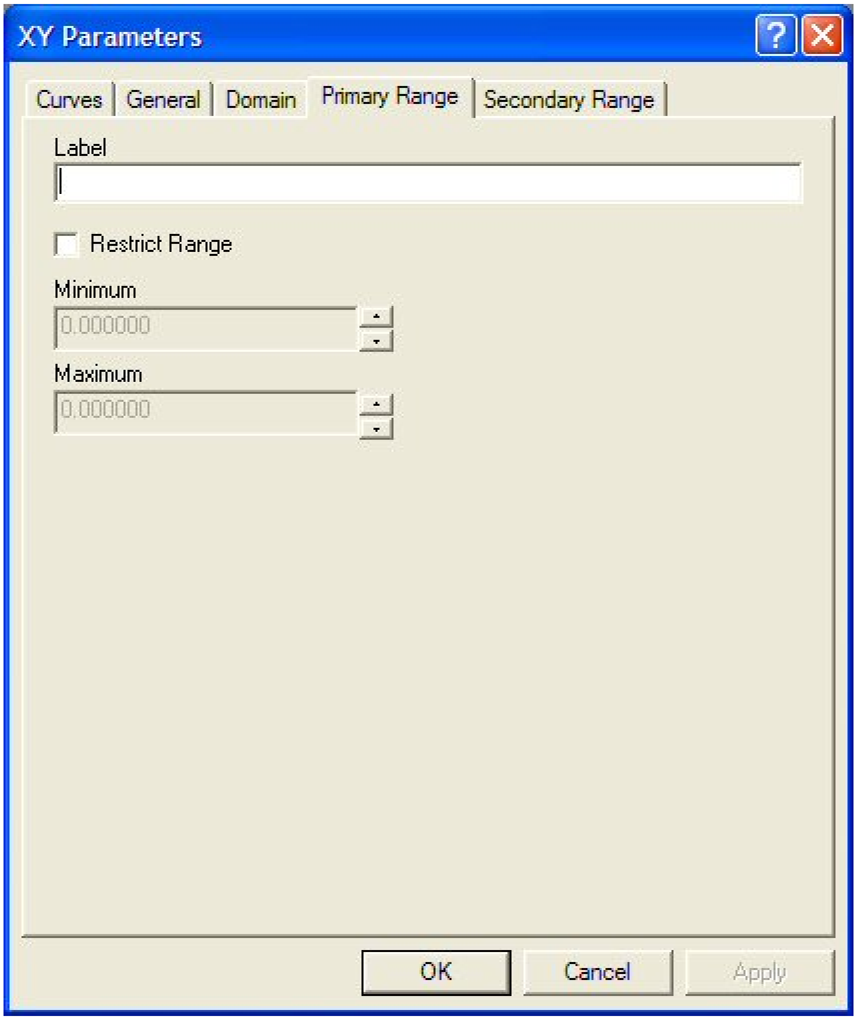


Figure 3-12 Y Axis Labeling

Secondary Range Label

This dialog box is used to describe the label that will be displayed above the y^2 axis on the graph. The y^2 axis is the one on the right hand side of the graph and is typically is used for concentration.

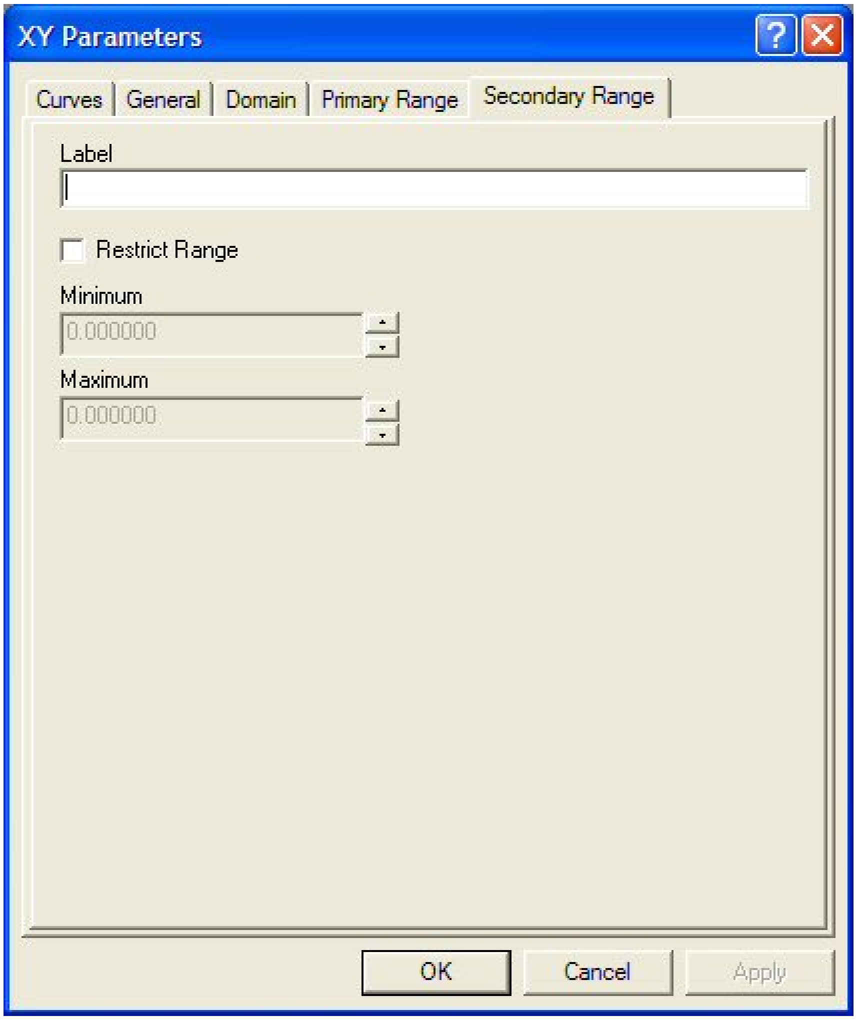


Figure 3-13 Secondary Y Axis Labeling

Time/Segment

This radio button option is used to inform that the x-axis domain is going to be used for time or distance. If the user selects distance, another option is available for the x-axis domain definition that is described below.

Map Segment to Distance

If the user selects the x domain to be of type segment, the user has the option of mapping the segment number to that of a river mile or distance. If this option is selected the user will need to create a database file that has the segment number with its corresponding distance. When this option is properly configured the distance and its corresponding concentration will appear on the x/y plot.

Restrict Domain

By default, the x-axis is automatically scaled using built-in heuristics based upon the range of the data. For the most part these heuristics provide very meaningful graphs. However the user has the ability to set the range of the x-axis manually. To manually control the range of x-axis, the user should select the "restrict x-axis range" from the graph configuration window. Once this is selected, the user may enter values for the minimum and maximum values for the x-axis.

> Note: There are more sophisticated methods for manipulating the range of the x-axis (see Zooming X-Axis).

Restrict y^1/y^2 Range

By default the y-axis is automatically scaled using built-in heuristics. For the most part these heuristics provide very meaningful graphs. However the user has the ability to set the scale of the y-axis manually. To enter a value for the y scale the user should select to restrict y-axis range from the graph configuration window. Once this is selected, the user may enter values for the minimum and maximum values for the y-axis being defined.

> Note: The user can control the range for both the Y^1 and Y^2 axes.

Graph Title

This dialog box is used to describe the title that will be displayed at the top of the graph. The user is limited to 25 characters within the title line.

Animation Delay

The animation delay dialog box allows the user to define the delay when the user presses the animation button from the x/y toolbar.

Color/Black & White

This radio button selection works the same as if the user pressed the Color/Black & White icon from the toolbar. This will toggle the x/y plot between color and black & white.

Curve Definition

The curves dialog window indicates how many lines are currently defined for the given graph. To add a curve to the graph the user should press the Add Curve button. This will bring up the curve definition window. If the user wants to edit the attributes of a previously created curve the user should select the curve to be edited in the curve window by clicking the mouse pointer on the curve and press edit curve. There is no limit to the number of curves that can be defined for a given x/y plot.

Okay/Cancel

The Okay/Cancel button determines what state the user wants to leave the form. If anything was modified in the plot parameter screen and the user selects Okay, the x/y plot window will be updated with the newly entered/selected information. If the user selects Cancel, any entered information would be lost and the x/y plot window would remained unchanged.

Curve Attributes/Adding a Curve

If the Add Curve button is pressed in the graph configuration menu, a second dialog box appears to define the attributes of the curve. Defining those attributes will result is a single curve being added to the x/y plot.

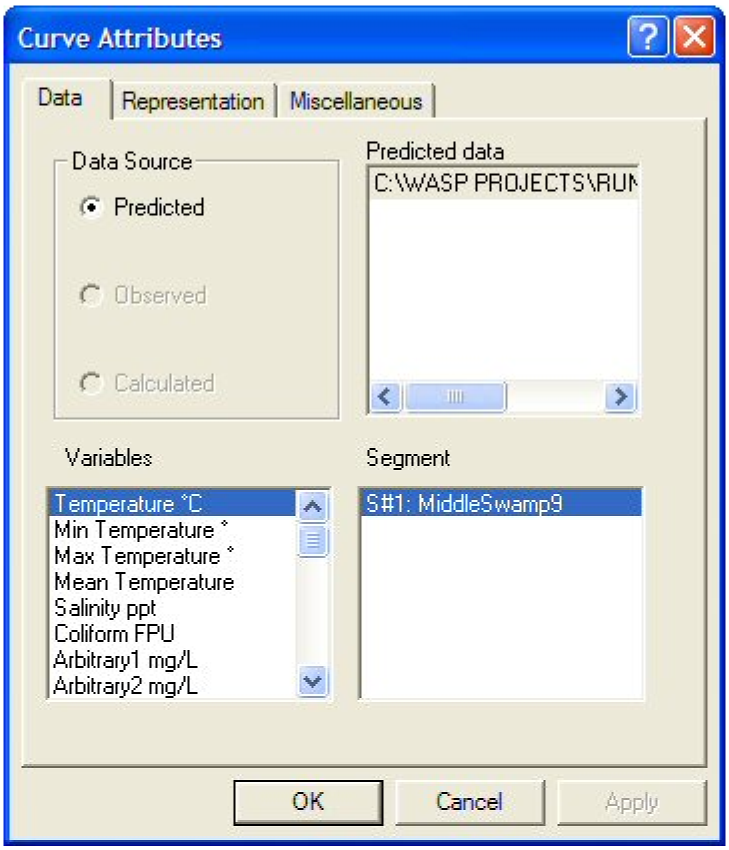


Figure 3-14 Input File Selection

Data Source

Data are obtained from several sources and are available for plotting within the x/y plot window. For the data to be available for a given plot window it must either be read into memory (read from disk: simulation result files or observed database) or created from a previous calculation.

The radio buttons located within the Data Source dialog window will be available for selection if the particular type is available for plotting. The model simulation results file and the observed data databases are loaded using the "Open File" dialog from the main menu. The calculated data sources are created from within the x/y plot window. A calculated data source created in one x/y plot window is available to other x/y plot windows as well.

Predicted

The predicted data type is assigned to the files that contain model simulation results.

Observed

The observed type is assigned to the observed data database. The observed data database must have a specific file format containing specifically named data fields.

Calculated

The user created the “calculated” field using built-in functions. One such function is the model partition function, where the user can calculate the difference between one curve and another. The results of this calculation are stored in a user defined Calculated data structure.

Selecting Data File

The selection of the data source radio buttons will determine the content of the data file dialog window. If a model simulation result file is loaded and the Predicted radio button is selected the filenames will appear in the Predicted Data window. The user selects the file to obtain plotting information from by pressing the left mouse button on the filename. The selected file will become highlighted. Once the file has been selected the user needs to select the variable and segment from which to retrieve information for plotting.

Selecting Variable

The variable list that is displayed in the variables dialog window is taken directly from the selected file. If it is a model simulation result file it will contain the output variables of either the hydrodynamic or water quality model.

If the selected file is an "observed data" database, the variables are the names of stored constituents in the database. If the selected file was a calculated data structure, the name of the variable will be that assigned by the user when the calculated field was created.

To select and view the variables in list the user can use the scroll bar to move through the list. To select a variable to plot, the user should click the left mouse button on the desired variable to select. The selected variable will become highlighted.

Selecting Segment

The last item that has to be selected before an x/y plot can be created is what segment or computational element to plot. Once a Predicted Data file is selected, a list of the available segments to plot is displayed in the Segment dialog window. To select and view the segments in the list the user can use the scroll bar to move through the list. To select a segment to plot, the user should click the left mouse button on the desired segment. The selected variable will become highlighted. If the data source is an observed data database the segment is typically a station identification number.

Representation

The representation radio button dialog box allows the user to assign a specific characteristic for the defined curve. The user has the option of drawing a line, symbol, line & symbol or force solid for the defined curve.

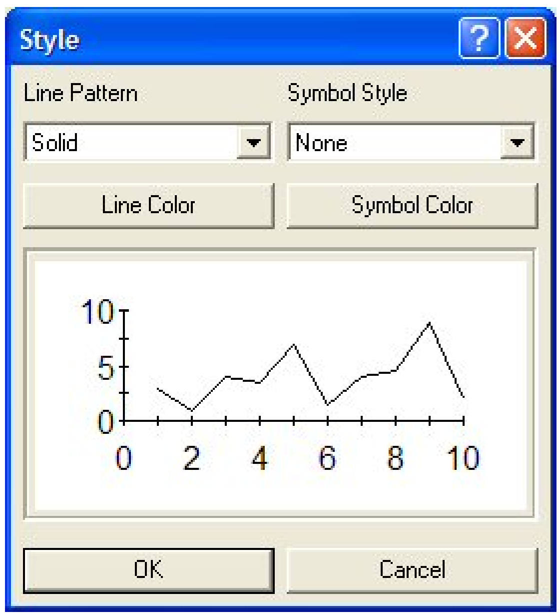


Figure 3-15 MOVEM x/y Plot Curve Attributes (User Defined)

> Note: A different line style, color and symbol is automatically created for each line defined in a particular x/y plot window.

Second Y

The user may assign any number of the defined curves to the second y^2 axis (scale located on the right hand side of graph). When a second y^2 is requested, the curve assigned to that y^2 axis will be scaled and plotted independently of any curve assigned to y^1 axis.

Observed Data

Observed data can be added to any x/y plot window (Figure 3-16). Before observed data is available for plotting the observed data database must have been opened for use by the user. Observed data is opened using the open file dialog box. Once the observed data database has been opened the data will be available for plotting in the x/y plot window. To select observe data to plot, the user should press the Observed radio button. If there is more than one database file loaded, the user will be presented a choice of file in the predicted data window. The user should select the database file, select the variable from the variable window and then select the Station ID from the segment window.

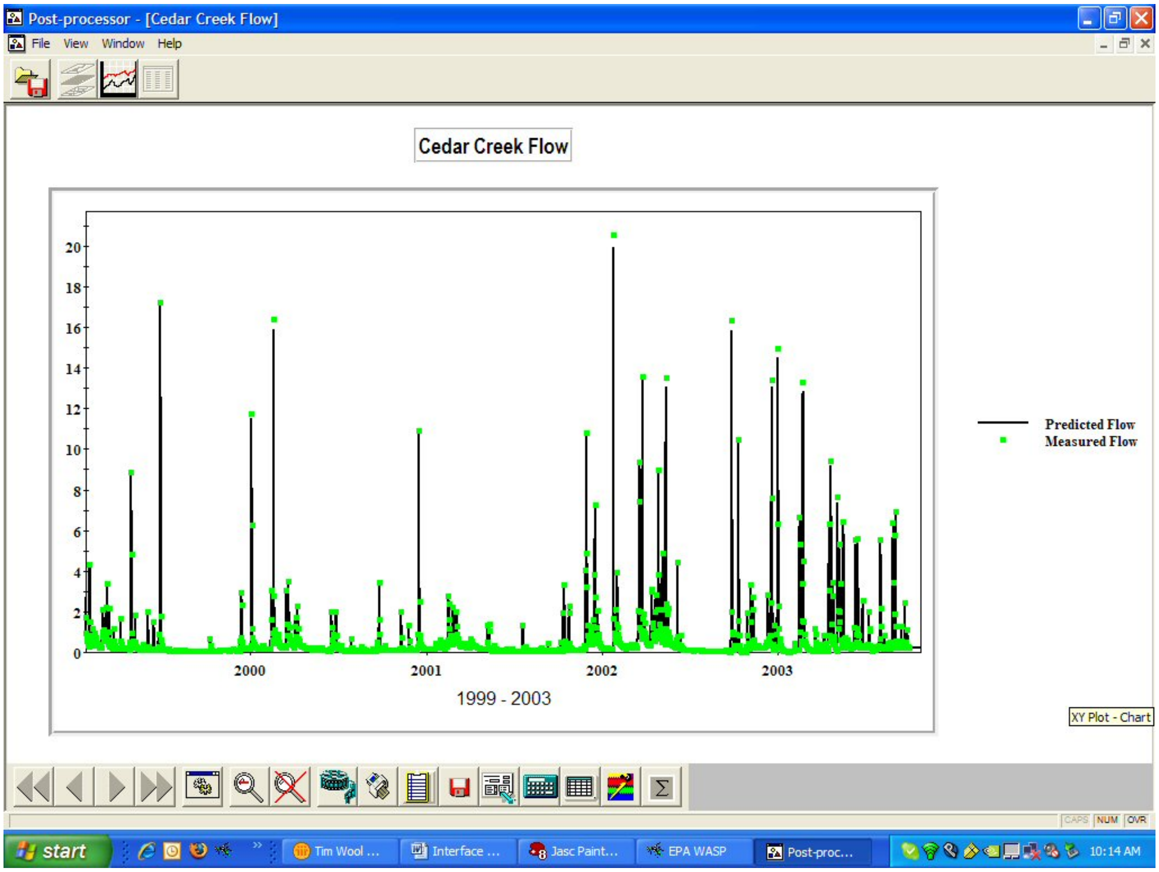


Figure 3-16 Plotting Observed Data versus Predicted

> Note: Observed data can be plotted exclusively if the user desires.

Calibration Statistics

The user has the ability to get calibration statistics from MOVEM. The calibration statistic option is only available when the user has one model prediction curve and one observed data curve defined for the currently x/y plot window. If this condition is met, the Calibration Statistic icon will be highlighted (i.e. available) on the x/y plot toolbar. Clicking on the calibration statistic icon will cause the calibration statistics to be generated. The user will have to give a filename in which additional calibration information as well as percentile information will be stored. Figure 3-17 illustrates the calibration statistic screen.

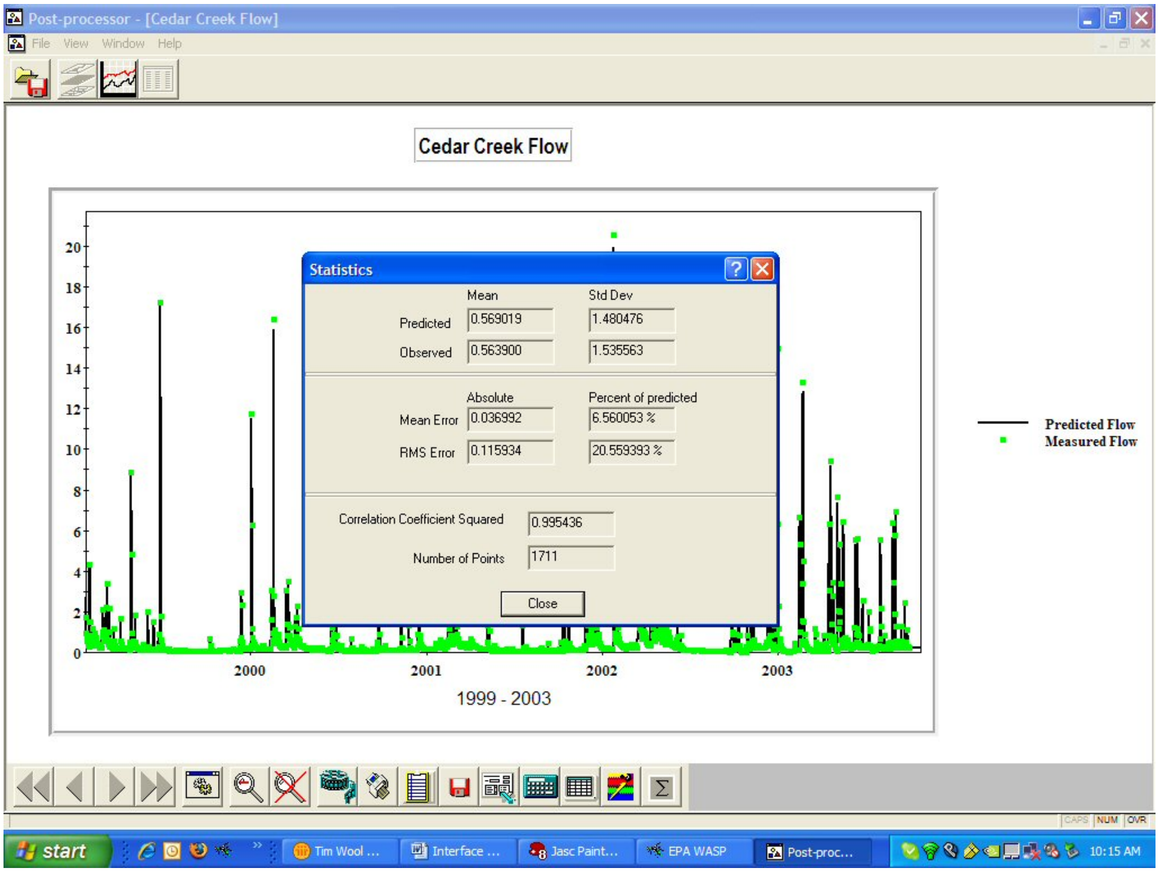


Figure 3-17 MOVEM Calibration Statistics

Legend Description

The user can control the information that is automatically placed in the x/y plot legend that is used to illustrate the contents of the graph. The user has four options for the legend:

1) Data Set – setting this radio button will cause the use of the dataset name selected in the Predicted Data dialog window for the legend of the defined line.

2) Variable -- setting this radio button will cause the use of the variable name selected in the Variable dialog window for the legend of the defined line.

3) Segment -- setting this radio button will cause the use of the segment name selected in the Segment dialog window for the legend of the defined line.

4) User-Defined -- setting this radio button will cause the use of the user-defined item to be placed in the User-Defined Dialog box.

> Note: If the user does not want a legend, the user should select "user define" and not type anything into the user defined dialog box.

3.4.3. OK/Cancel

Once this information was been selected along with any other user definable parameters the user should press Okay to generate the graph. The observed data will be plotted on the x/y graph along with any predicted or user selected data. If they press cancel, all the information entered will be lost.

3.4.4. Zooming the Axes

The user can zoom any of the axes within the x/y plot window. The user can zoom the x-axis, y^1-axis and y^2-axis exclusively or in combination. Zooming the axis allows the user to view data in smaller time or concentration scales to visualize subtle changes in the model results.

Zooming X Axis

The user has several options available performing a zoom function. The quickest and most efficient way to zoom the axis is to place the mouse cursor close to the x-axis line (within the plot area) at the beginning time of the area to zoom. Then press and hold the left mouse and paint the area to zoom by dragging a box to the end time. The zoomed area will be painted.

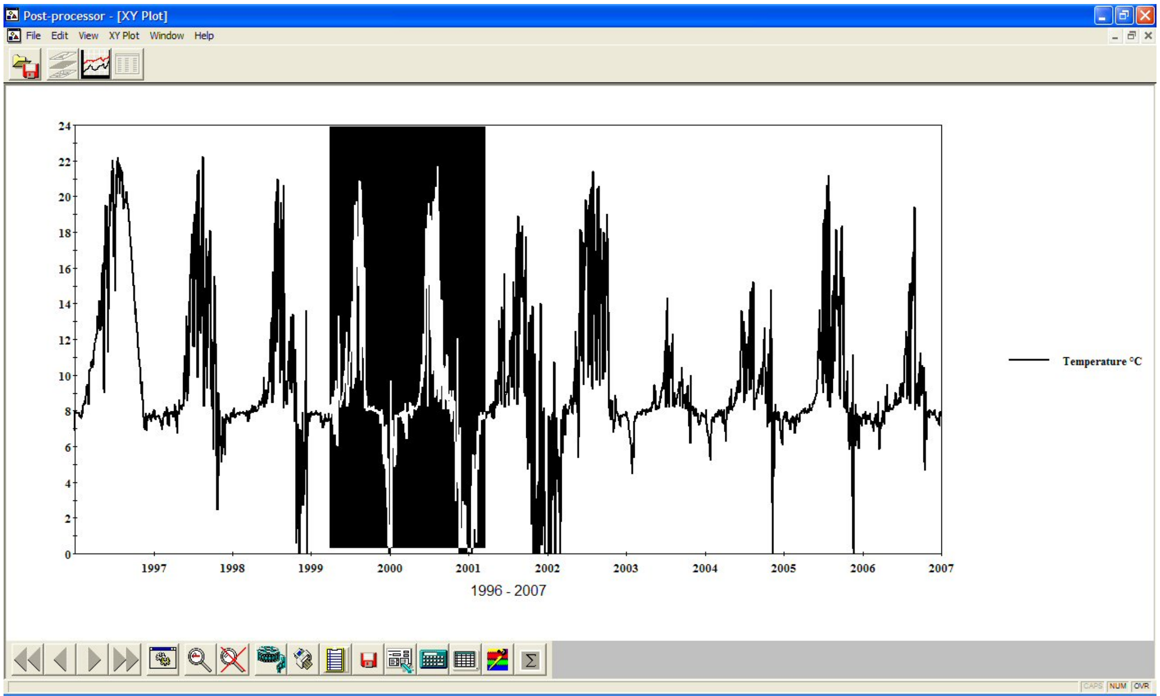


Figure 3-18 Zooming the x Axis

Upon releasing the left mouse button, after the user has defined the area to zoom, the x/y plot will be redrawn for the given time period selected by the user. Figure 3-19 illustrates a zoomed x-axis, from the example given above.

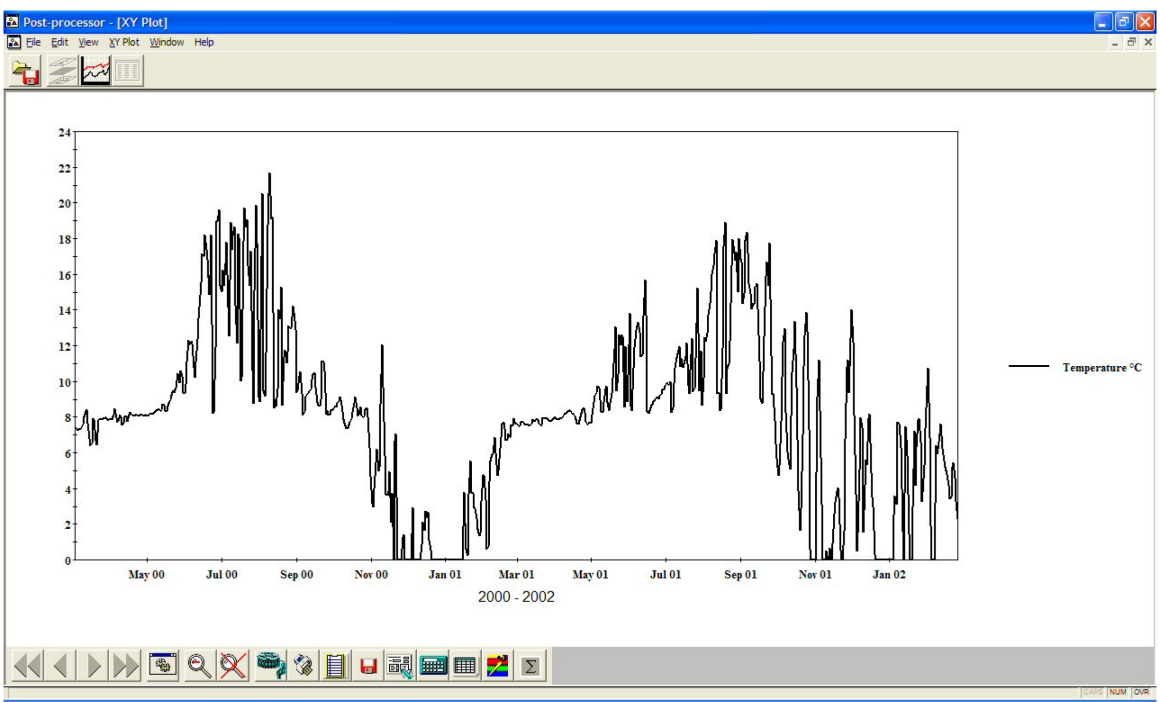


Figure 3-19 Zoomed x Axis

> Note: The user can zoom the axis in steps (i.e. zoom look at the graph and then zoom the zoomed graph some more). The user also has two other functions available from the toolbar: un-zoom x/y plot, or return to previous zoomed level.

Zooming the Y

The y-axes may also be zoomed. The user has several options available to indicate the desire to perform a zoom function. If only one y-axis has been defined, the axis can be zoomed just like the X. To zoom the y-axis, place the mouse cursor close to the y-axis line (within the plot area) at the beginning concentration of the area. To zoom, press and hold the right mouse and paint the area to zoom by dragging a box to the end concentration (Figure 3-20). The area that will be zoomed will be painted.

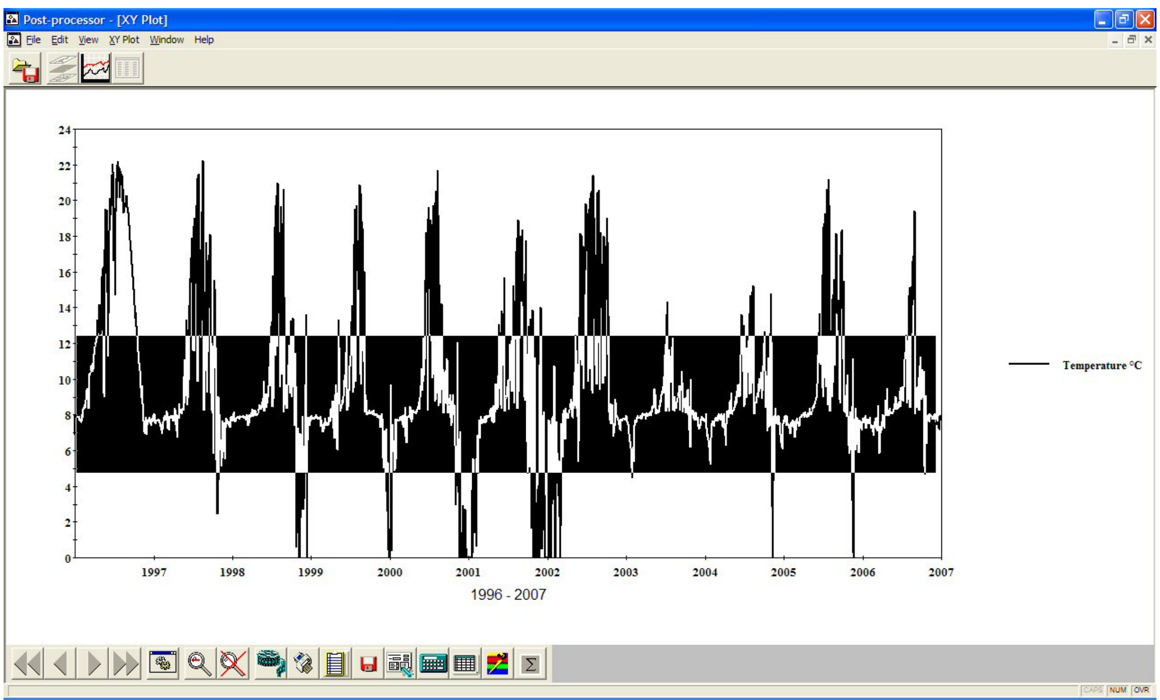


Figure 3-20 Zooming the y Axis

Upon releasing the right mouse button, after the user has defined the area to zoom, the x/y plot will be redrawn for the given y concentration range. Figure 3-21 illustrates a zoomed y-axis, from the example given above.

If the user has defined more than one y-axis, the zooming function becomes a little trickier to perform. If the zooming function on the speed menu is set to Auto-Detect (the default), the Graphical Post-Processor will determine which axes (x, y^1, or y^2) the user is trying to zoom based upon the position of the mouse at the time the left mouse button is pressed. If the user is closer to the Y^1 axis and presses the right mouse button and drags a box vertically, it assumes the user wants to zoom the Y^1 axis. The converse is true for the

y^2 axis. If the user is closer to the x-axis and drags a box horizontally, it is assumed the user wants to zoom the x-axis.

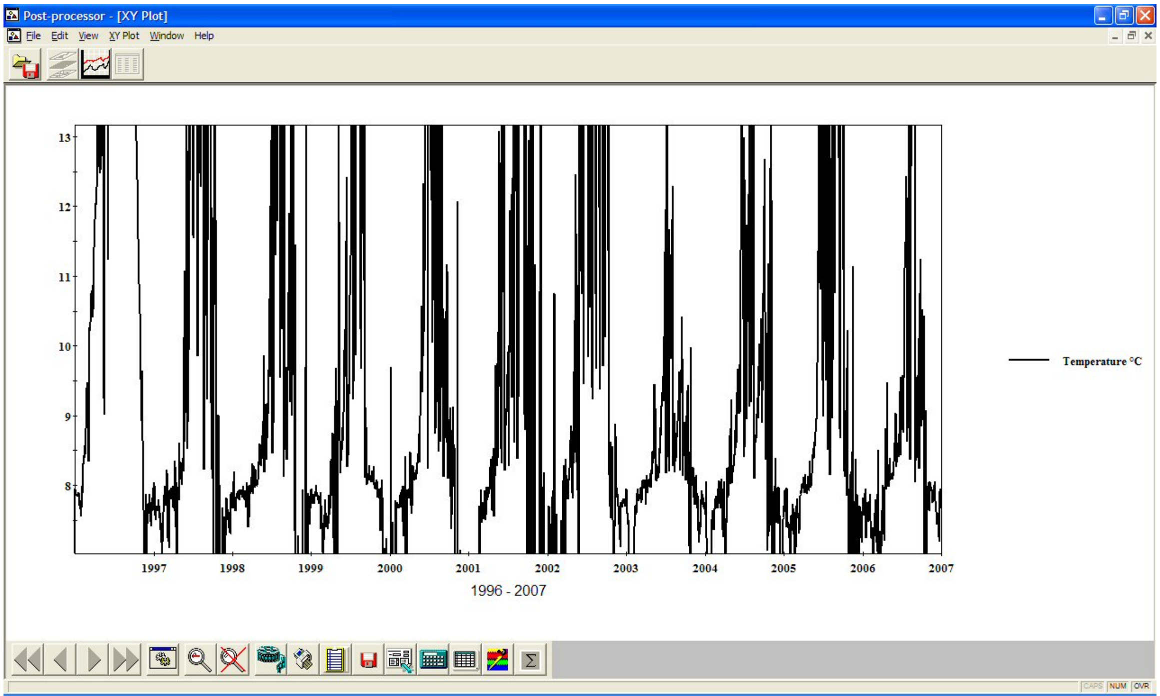


Figure 3-21 Zoomed y Axis

> Note: If the user is having trouble controlling which axis to zoom, the speed menu allows the user to select which axis to zoom.

3.4.5. Adding an Additional Curve

The user has the option of adding as many curves as desired to any given x/y plot window. There is no limit to the number of curves that can be defined. It is recommended that no more than five curves be defined for a given x/y plot as resolution and comprehension will be diminished.

> Note: The user can create as many curves per x/y plot and x/y plot windows as desired.

3.4.6. Color/Black & White View

The x/y plots can either be displayed in black & white or color depending upon user preference or intentions. To toggle between these two modes the user can press the black and white icon from the toolbar, toggle the radio box in the configuration menu or use the speed menu. An example of a black & white plot is given in Figure 3-22. When the user requests that a black & white graph be printed or copied to the clipboard, a black & white image is generated.

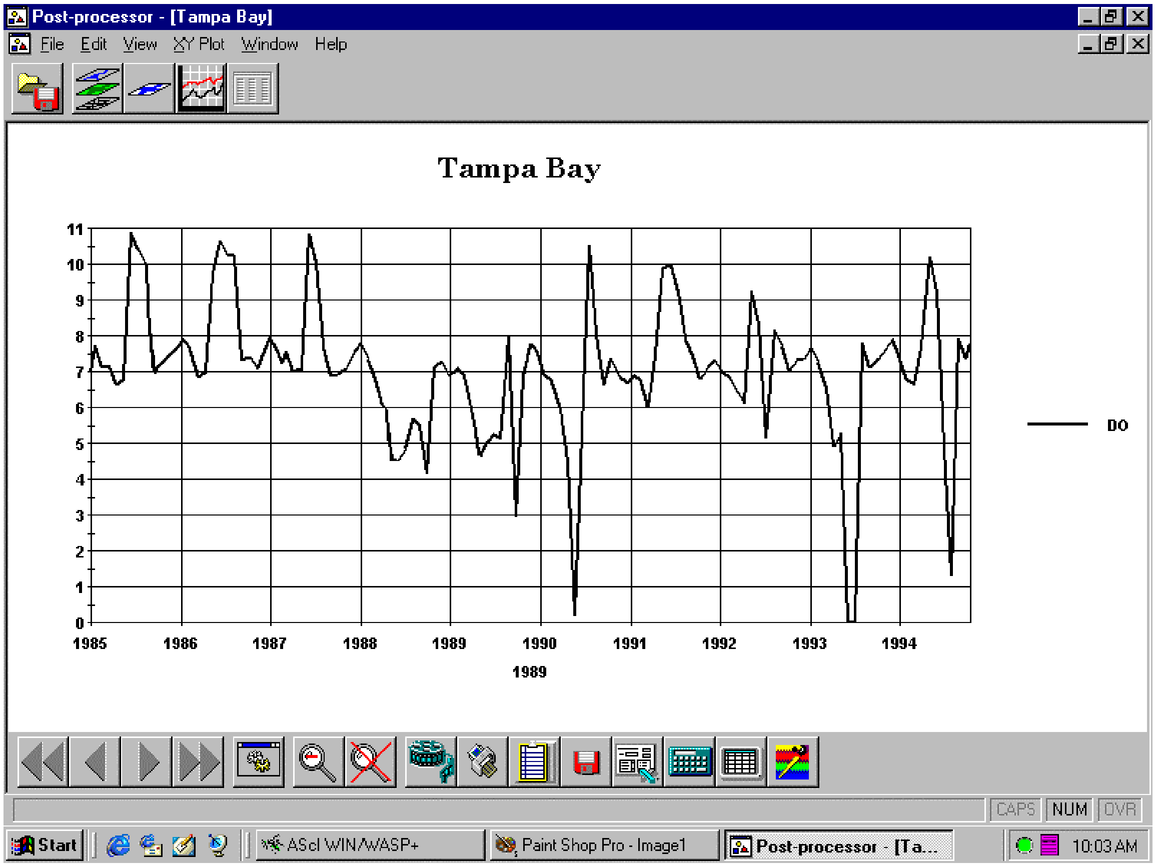


Figure 3-22 Example of Black & White Graph

3.4.7. Observed/Measured Data

Overview

The user may plot observed/measured data against that predicted by the model. Observed data have to be stored in a particular file format to be available for plotting. The file formats are Paradox 4.5 or higher database tables. Also, the Paradox database must contain at least four requisite field names. These field names are used to align the data from the database to the dialog boxes of the x/y plot curve parameters.

Creating a Database

The user has several options available for creating the observed data database. If the user has Paradox they can use it to create the observed data database. If the user does not have Paradox, the user is provided the option of creating an observed data database. If the user elects to create the observed data database in Paradox there are four important fields that the database must contain:

1) DateTime – this field is of type Timestamp. It is used to store the date & time of the observed data point.

2) PCODE – this field is of type alphanumeric. It is used to describe the type of measurement being stored (i.e. Dissolved Oxygen).

3) STATION_ID -- this field is of type alphanumeric. It is used to describe the sample station identification.

4) RESULT -- this field is of type numeric. It is used to describe the measured value of PCODE at time DATETIME at STATION_ID.

The four fields above have to be defined exactly as described to be useful. Any variance from what is given above and the file will not be recognized. The observed data database can contain more than the four fields described above, but at a minimum must contain these fields.

To create a new observed data database the user should press the Create New Database icon on the main toolbar. This will cause a new observed data database table to be generated that the user can populate with their own data. Figure 3-23 illustrates a newly created observed data database table. The user must populate the database with their own data by using the insert record function (pressing the + sign). The user can paste data into this table from other applications as well. To do this the user needs to insert as many records that they will be pasting into the table.

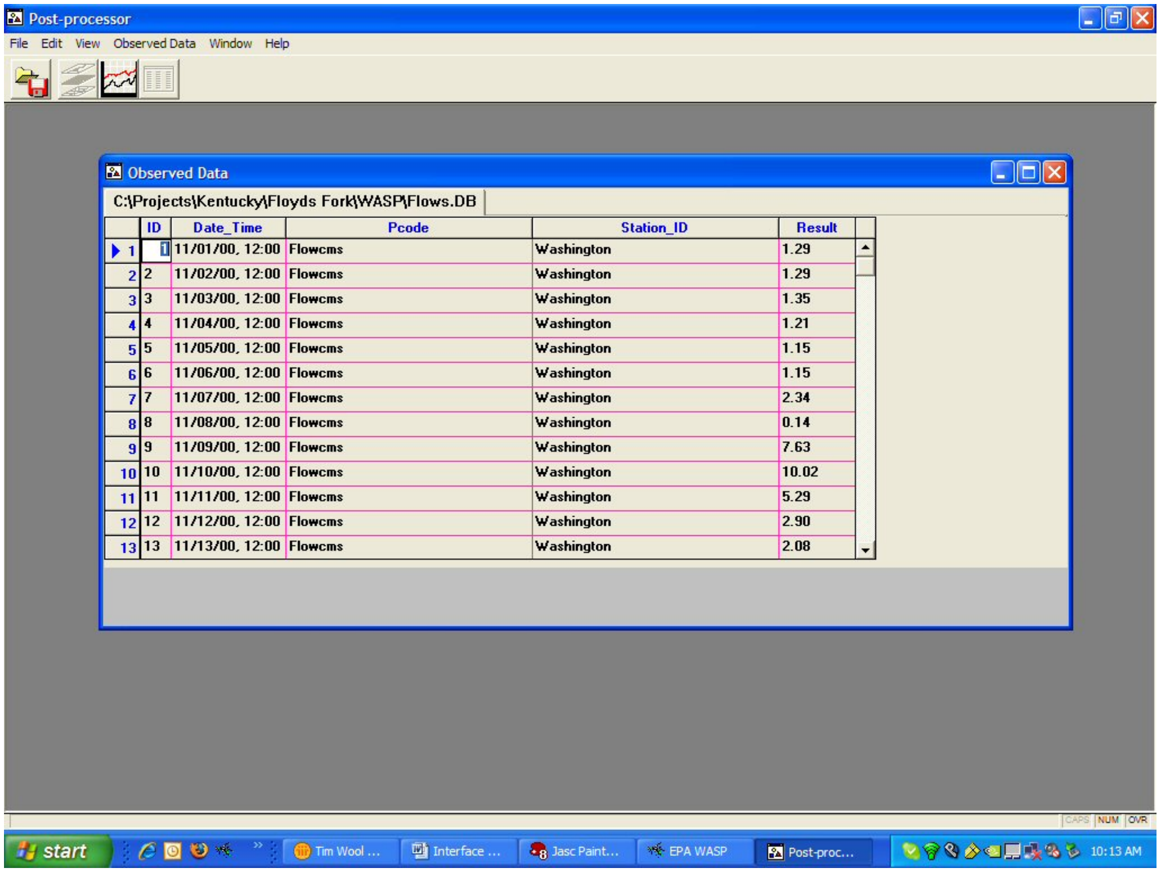


Figure 3-23 Creating Observed Data Database

Loading a Database

Once an observed data database has been created it must be loaded like any other file before it is available for plotting. To load an observed data database file, the user should select the open file icon or select it from the file menu. Upon doing this a file dialog box will appear, the user should set the file type to that of *.DB. The user should navigate to the drive and directory where the observed data database is stored and select the file and press open. Once the contents of the observed data database are read into memory they are available for plotting.

Selecting Data

Selecting data from the observed data database file is done in the same manner as selecting data from the model simulation results files. The user should select Add Curve from the x/y plot configuration menu, when the curve attribute menu appears the user should select the Observed Data radio button. Once the radio button has been activated the previously loaded DB file name should appear in the data file dialog box. The user should highlight the DB filename with the mouse, select the database PCODE and Station_ID to plot. Once these items have been selected the user should press the Okay button and the graph will be re-drawn with observed data included.

> Note: The user has the same control over the observed data's appearance as the simulation results.

3.4.8. Printing Results

The user has the option of printing the currently active x/y plot to any installed Windows printer device (this could be a laser printer, color printer, fax, or even e-mail). The user has the option of either printing the x/y plots in full color or changing them to black and white before printing. Depending upon the printer and the users Window's 95 setup, the printer may automatically convert the colors to gray scale before printing. This may or may not be advantageous, if the graphs appear "muddy" the user should convert the graphs to black and white before printing to a printer, saving to a file or copying to a clipboard.

To change a plot to black and white the user should either select the Black and White radio button from the configuration menu or press the Black & White/Color toggle icon from the toolbar.

To print the currently active x/y plot the user should press the print icon from the toolbar. A standard windows print dialog box will appear. The user can select the appropriate output device to print the figure.

To File

The user has the option of saving a bitmap file of the currently active x/y plot to a file. This is useful for saving simulation results for comparison or inclusion into a presentation or publication.

To save the currently active x/y plot to a file as a bitmap, the user should press the save graph to bitmap icon from the toolbar. The user will be given a standard windows file dialog box that allows the user to designate the drive, directory and filename.

To Clipboard

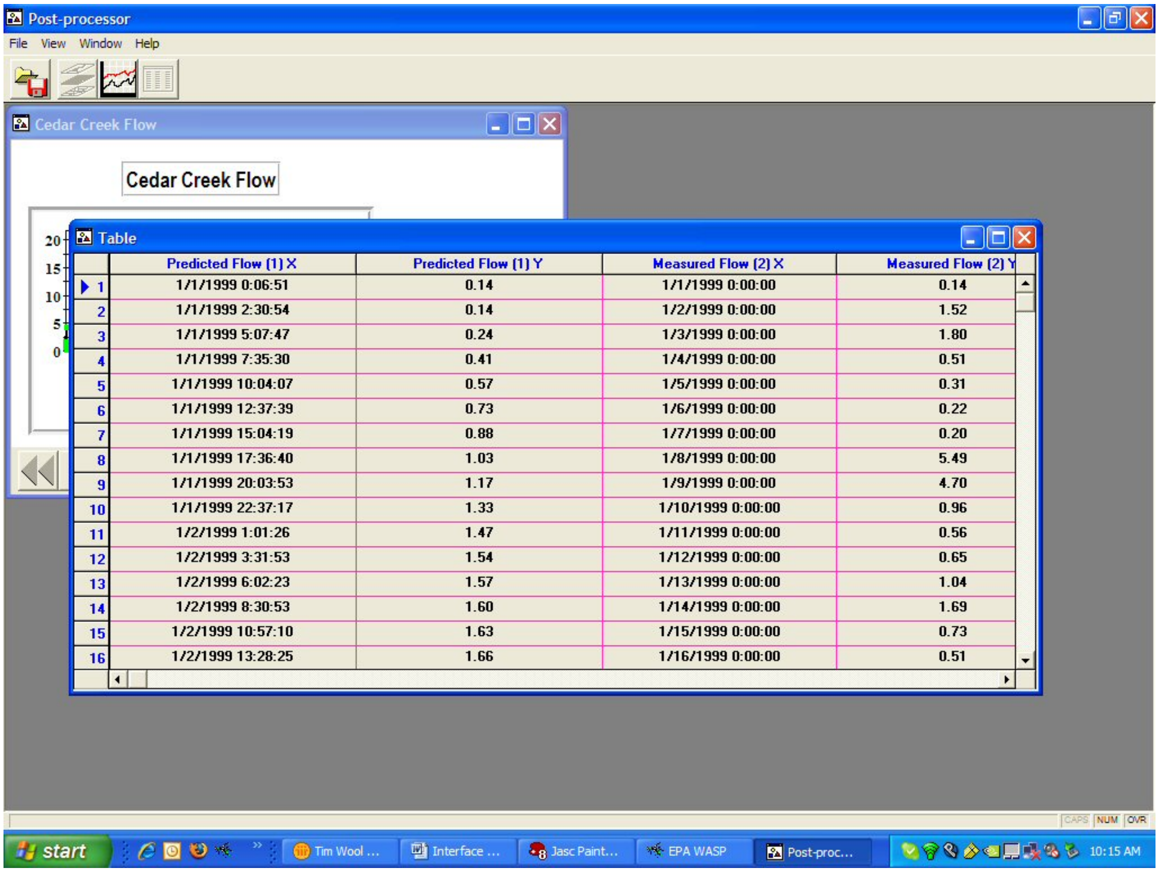
The user also has the ability to copy a graphic image of the currently active x/y plot to the Windows clipboard. Once an image is copied to the Windows clipboard it can be pasted into virtually any Windows program that has graphics support.

To copy the graphics image to the clipboard the user should press the copy to clipboard icon from the toolbar. Once the image has been copied it is available for pasting.

3.4.9. Creating Tabled Data

Internal data structures are utilized to store the actual numbers that are used to generate the x/y plots. A method is provided for the user to create a table of the data that is represented in the current x/y plot. Pressing the data table icon on the x/y plot toolbar creates this table. This will cause the creation of a table of data as illustrated in Figure

3-24. The data in this table is "read-only". This basically means the user can access the data by marking columns and rows of data and copying it to the Windows clipboard so that it can be pasted into another application (i.e. spreadsheet).



	Predicted Flow (1) X	Predicted Flow (1) Y	Measured Flow (2) X	Measured Flow (2) Y
1	1/1/1999 0:06:51	0.14	1/1/1999 0:00:00	0.14
2	1/1/1999 2:30:54	0.14	1/2/1999 0:00:00	1.52
3	1/1/1999 5:07:47	0.24	1/3/1999 0:00:00	1.80
4	1/1/1999 7:35:30	0.41	1/4/1999 0:00:00	0.51
5	1/1/1999 10:04:07	0.57	1/5/1999 0:00:00	0.31
6	1/1/1999 12:37:39	0.73	1/6/1999 0:00:00	0.22
7	1/1/1999 15:04:19	0.88	1/7/1999 0:00:00	0.20
8	1/1/1999 17:36:40	1.03	1/8/1999 0:00:00	5.49
9	1/1/1999 20:03:53	1.17	1/9/1999 0:00:00	4.70
10	1/1/1999 22:37:17	1.33	1/10/1999 0:00:00	0.96
11	1/2/1999 1:01:26	1.47	1/11/1999 0:00:00	0.56
12	1/2/1999 3:31:53	1.54	1/12/1999 0:00:00	0.65
13	1/2/1999 6:02:23	1.57	1/13/1999 0:00:00	1.04
14	1/2/1999 8:30:53	1.60	1/14/1999 0:00:00	1.69
15	1/2/1999 10:57:10	1.63	1/15/1999 0:00:00	0.73
16	1/2/1999 13:28:25	1.66	1/16/1999 0:00:00	0.51

Figure 3-24 Example of Tabular Data from Graph

The user has no ability to modify the data and have the results appear in the x/y plot.

Exporting Data

The user may export the data that is used to generate the active x/y plot to an external file to 1) comma delimited ASCII files and 2) Paradox database files.

The comma delimited ASCII file type writes the data from the graph in column format, each curve will consist of two columns (x & y values). The data is separated by commas and can be directly imported into most spreadsheet programs.

The Paradox file format creates a Paradox compatible database file of all the data in the x/y plot. This database table that is created can be read directly by Paradox or the Deliberator program (a component of WASP). Each x/y pair from the graph is given it's own record in the database.

To initiate the export data option the user should press the export data icon on the x/y plot toolbar. This will bring up a file dialog box that allows the user to define the path and filename to save the exported data.

To select the file type in which to save the data the user should use the drop down list from the Save As type in the dialog box. Chose the appropriate type and give the file a name and press save.

3.4.10. Curve Calculations

A wide range of calculations can be performed on defined curves on the x/y plot. The curve calculation screen is entered by pressing the curve calculate button on the x/y plot toolbar. Curve calculations can only be performed on defined curves within the x/y plot window. The are several types of curve calculations that can be performed:

1) User Defined – functions that the user can derive to make calculation

2) Moving Average – user specifies the time interval for the moving average and generates a new curve depicting the results.

3) Frequency Distribution – the user specifies the number of concentration intervals to be used in the frequency distribution and calculation of the new curves.

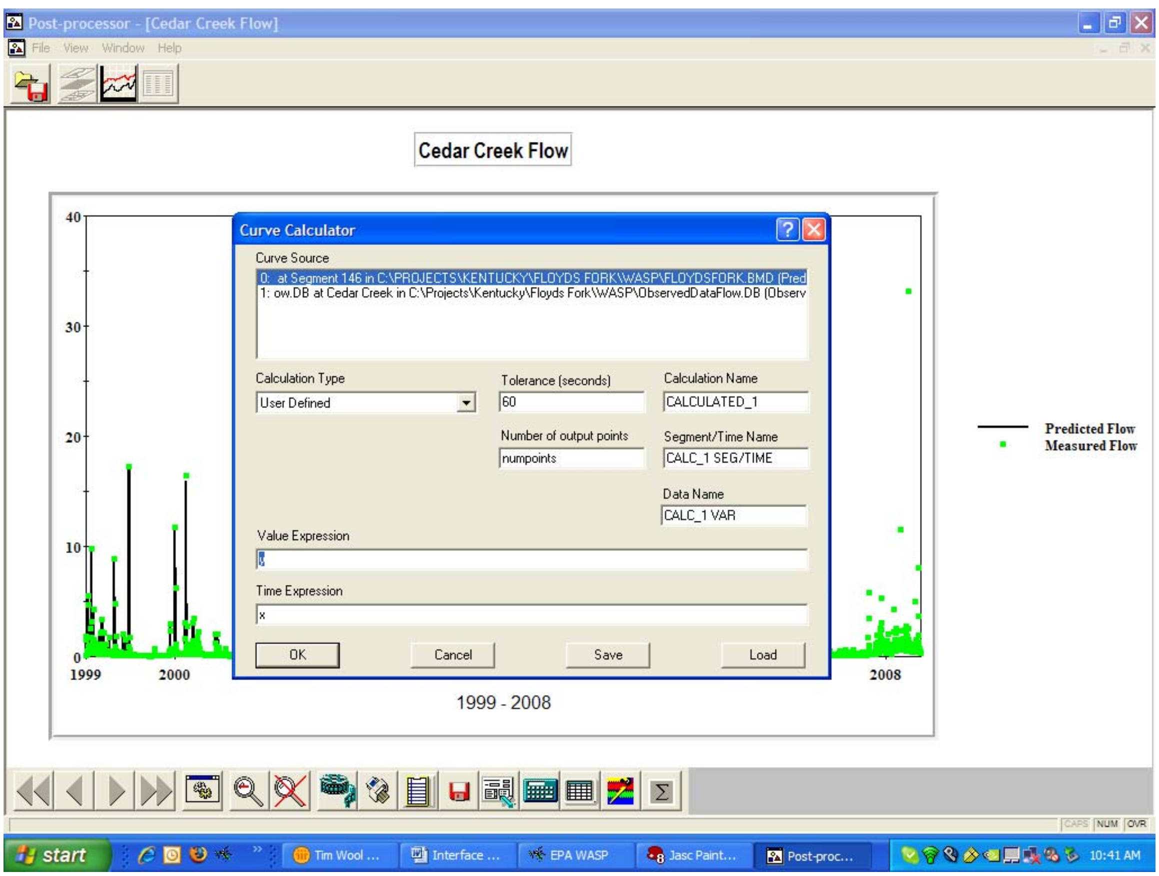


Figure 3-25 Example of Curve Calculation

The curve calculation screen has several dialog boxes that provide information to the user as well as allow the user to specify information and operations.

Curve Source

The curve source dialog allows the user to select which loaded data source will be used for defining a curve.

Calculation Type

The user has the option of selecting the type of calculation that should be performed. The calculation type is selected from a drop down picklist as illustrated in Figure 3-26. The user has the option of user defined, moving average or frequency distribution.

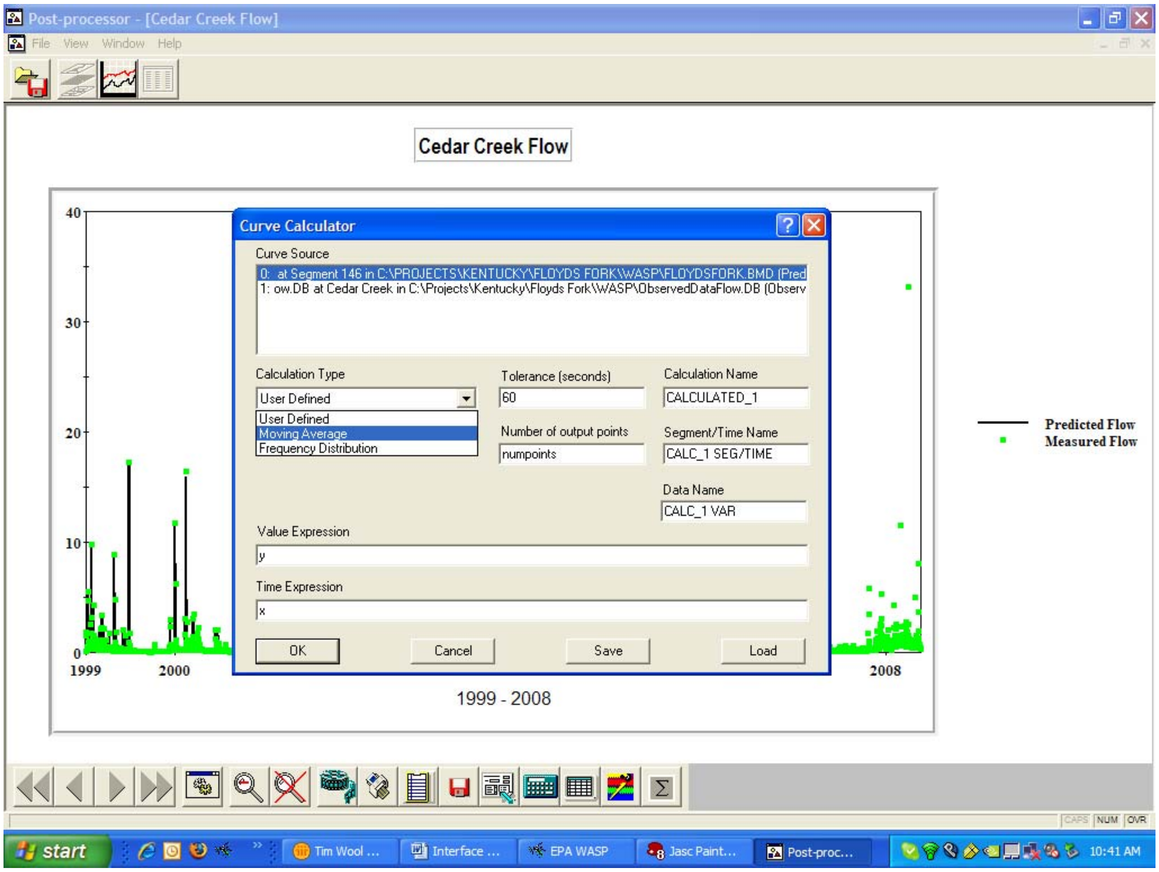


Figure 3-26 Built in Curve Calculation Functions

Tolerance

The tolerance factor is used to sets the range of time that would consider point to be at the same time.

Number of Output Points

Defines the number of output points that will be produced for the curve calculations.

Calculation Names

The calculation name dialog box provides a means for the user to give a meaningful name to the calculation. This is an important step, because once the calculation is made, to display the calculation in the x/y plot the user will need to add a curve to the current or new x/y plot window. To do this, the user would select the calculate type, from the curve parameter window. Once this is selected a picklist is provided of available calculations. This picklist contains the calculation name as defined by the user.

Segment Time/Number

These dialog box sets the label displayed on the graph for the segment/time axis.

Data Name

This dialog box sets the label for the value in the x/y plot legend.

Value Expression

This dialog box is used to define the function that will be performed on the values.

Time Expression

This dialog box is used to define the function that will be performed on the time values.

The following tables provide information for the pre-defined variables and functions that are available.

Variable	Pre-Defined Variables
X	Value of cell being calculated or X-value of curve point being calculated
Y	Y-value of curve point being calculated
R	Row number (1 - Number of rows) of cell being calculated
C	Column number (1 - Number of columns) of cell being calculated
NUMPOINTS	Returns the number of points in the reference curve.
I	Current point number being calculated
REFCURVE	Curve Selected in List box of Curve Calculator
PI	PI (One half the circumference of a unit circle : 3.141592)
SECSPERDAY	Number of seconds in a day (86400)
NUMCURVES	Number of Curves on Graph
E	Base of Natural Log
CELL(row, column)	Value of cell located at (row, column)
ABS(x)	Absolute value of the expression x
FLOOR(x)	The largest integer not greater than the expression x
CEIL(x)	The smallest integer not less than the expression x
POW(x, y)	x raised to the y power
SQRT(x)	The square root of the expression x
FT3M3(x)	Conversion of the expression x from cubic feet to cubic meters
M3FT3(x)	Conversion of the expression x from cubic meters to cubic feet
FTM(x)	Conversion of the expression x from feet to meters
MFT(x)	Conversion of the expression x from meters to feet

FT2M2(x)	Conversion of the expression x from square feet to square meters
M2FT2(x)	Conversion of the expression x from square meters to square feet
MGDCFS(x)	Conversion of the expression x from million-gals/day to cubic feet/sec
SIN(x)	Sin of x (where x is in radians)
COS(x)	Cos of x (where x is in radians)
LOG(x)	Log of x
LN(x)	Natural Log of x
CFSGM3LBSDAY(x)	Converts flow (cfs) * concentration (g/m3) to Lbs/day
CFSGM3KGDAY(x)	Converts flow (cfs) * concentration (g/m3) to kg/day
CMSGM3KGDAY(x)	Converts flow (cms) * concentration (g/m3) to kg/day
MAX(c)	Maximum Number in the Curve
MIN(c)	Minimum Number in the Curve
STDDEV(c)	Standard Deviation of Curve
AVG(c)	Average Value for Curve
MEDIAN(c)	Median Value for Curve
MEAN(c)	Mean value for Curve
VARIANCE(c)	Variance Value for Curve
MODE(c)	Mode value for curve
X(c)	X value of any curve at current point
Y(c)	Y value of any curve at current point
NUMPOINTS(c)	Number of Points in Curve
X(c, p)	X value of any curve at any point
Y(c, p)	Y value of any curve at any point
INTERPOLATE(c, x)	Interpolates the value of curve c at domain value x
DATE(m, d, y, h, m, s)	Returns x value for a given date
SUM(c)	Sum of the values in the curve

Frequency/Running Average

These are built-in calculations. They work much the same as the user defined with the exception the user does not have to enter any functions, they are done automatically.

Once this dialog box has been completed and the user has pressed the Okay button, this partition calculation can now be plotted. To plot the partition calculation the user should press the Add Curve button. The curve parameter dialog box will appear. The user should select the Calculate radio button. Once Calculate is selected the user should see the name of the calculation just performed appear in the file dialog box. The user can then select the variable and location to plot. Once these are selected and the user presses "Okay", the x/y plot window will re-appear with the calculated data plotted as well.