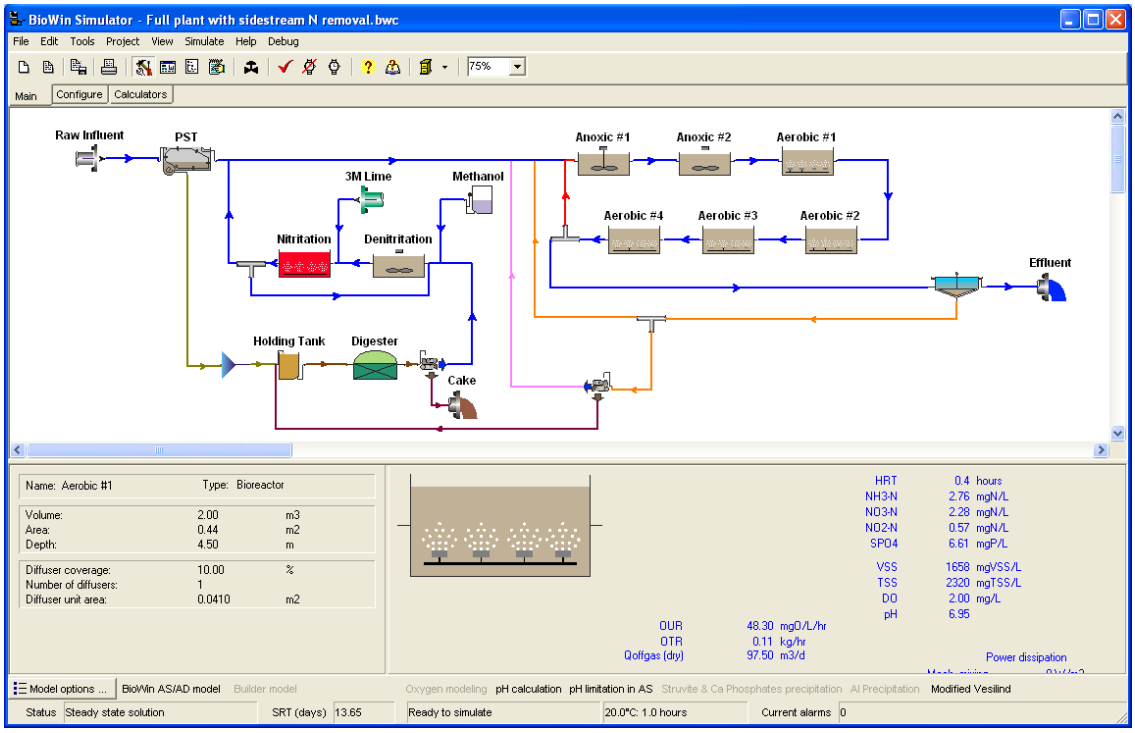
User Manual For

BioWin 3

EnviroSim Associates Ltd.



EnviroSim License Agreement

This is a legal agreement between you (either an individual or an entity) and EnviroSim Associates Ltd. If you do not agree with the terms of this agreement, return the software and the accompanying material to EnviroSim.

1. GRANT OF LICENSE. EnviroSim grants you the right to use one copy of the enclosed EnviroSim software program on a single computer. You may not network this software or otherwise use it on more than one computer at the same time.

2. COPYRIGHT. This software is owned by EnviroSim and is protected by Canadian copyright laws and international treaty provisions. Therefore, you must treat the software like any other copyrighted material (for example, a book), with the following single exception. EnviroSim authorizes you to make copies of the software so long as there is no possibility of it being used on more than one computer at the same time. You may not copy the manual(s) or written materials accompanying the software.

3. LICENSE VALIDITY. The software verifies the license validity using a hardware-based software protection device, hereafter called the HardLock Key. This HardLock Key must be connected directly to a single PC on which the software is installed to execute the software.

4. LEASE CONDITIONS. The GRANT OF LICENSE is limited to the period for which the LEASE is valid. The HardLock Key remains the property of EnviroSim and must be returned to EnviroSim if the lease is not renewed.

5. OTHER RESTRICTIONS. You may not sell, rent or lease the software or any of the accompanying materials (including, but not limited to HardLock Key). You may not reverse engineer, decompile or disassemble the software.

Limited Warranty

EnviroSim warrants that (a) the software will perform substantially in accordance with the User Manual(s) for a period of 90 days from the date of receipt; and (b) any hardware accompanying the software will be free from defects in materials and workmanship under normal use and service for a period of one (1) year from the date of receipt. Any implied warranties on the software and hardware are limited to 90 days and one (1) year, respectively.

EnviroSim's entire liability and your only remedy shall be, at EnviroSim's option, either (a) return of the purchase price or (b) repair or replacement of the software or hardware that does not meet EnviroSim's Limited Warranty. This Limited Warranty is void if failure of the software or hardware has resulted from accident, abuse, or misapplication.

EnviroSim disclaims all other warranties, either express or implied, including but not limited to implied warranties of merchantability and fitness for a particular purpose, with respect to the software, the accompanying manual(s) and written materials, and any accompanying hardware.

In no event shall EnviroSim Associates Ltd. and/or it's owners be liable for any loss of profit, or any other commercial damage, including but not limited to special, incidental, consequential, or other damages arising out of the use of or inability to use this EnviroSim product.

EnviroSim's entire liability under any provision of this agreement shall be limited to the amount paid by you for the software.

Should you have any questions concerning this Agreement please contact:

EnviroSim Associates Ltd.,
7 Innovation Drive, Suite 205,
Flamborough, Ontario
L9H 7H9, CANADA.

Table of Contents

Welcome To BioWin

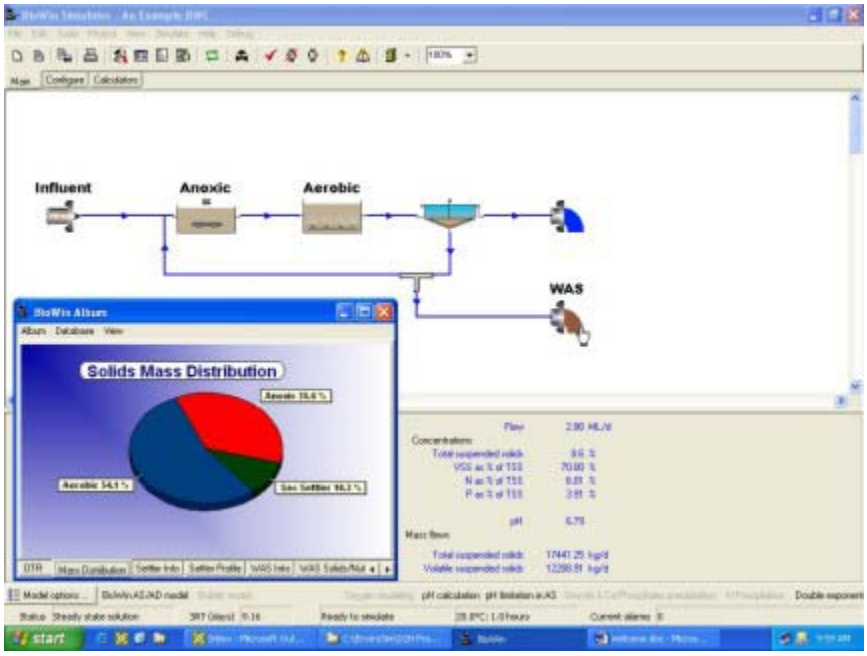
Welcome

Welcome to the latest version of BioWin – BioWin 3 - a comprehensive simulation tool for biological wastewater treatment plant design, analysis and training. The package was developed with the primary objective of providing a powerful tool to aid both the process designer and operators of these facilities.

For getting started, go to the "*BioWin Tutorials*". This set of tutorials and case studies is designed as a training exercise in the application of BioWin. The primary objective is to provide **"how to…"** training on using the BioWin software itself. The case studies are not intended as a course in wastewater treatment process engineering. Nevertheless, several of the case studies focus on process applications and identify interesting design and operating issues.

There are two places where you can find more BioWin examples:

- Select **File|Open** and browse to the **Examples** directory. These systems are discussed in the Help section on BioWin Tutorials and Examples.
- On the BioWin main window toolbar, at the end on the right, click on the arrow next to the icon that looks like a filing cabinet. This brings down a list of pre-configured BioWin process files for a range of system configurations.



The BioWin wastewater treatment process simulator

BioWin in Brief

The user can define and analyze behavior of complex treatment plant configurations with single or multiple wastewater inputs. An example of a plant configuration is shown below.

Most types of wastewater treatment systems can be configured in BioWin using the many process modules. These include:

- A range of activated sludge bioreactor modules – suspended growth reactors (diffused air or surface aeration), various SBRs, media reactors for IFAS and MBBR systems, variable volume reactors.
- Anaerobic and aerobic digesters.
- Various settling tank modules – primary, ideal and 1-D model settlers.
- Different input elements – wastewater influent (COD- or BOD-based), user-defined (state variable concentrations), metal addition for chemical phosphorus precipitation (ferric or alum), methanol for denitrification.
- Other process modules – holding tanks, equalization tanks, dewatering units, flow splitters and combiners.

A crucial component of BioWin is the biological process model. The BioWin model is unique in that it merges both activated sludge and anaerobic biological processes. Additionally, the model integrates pH and chemical phosphorus precipitation processes.

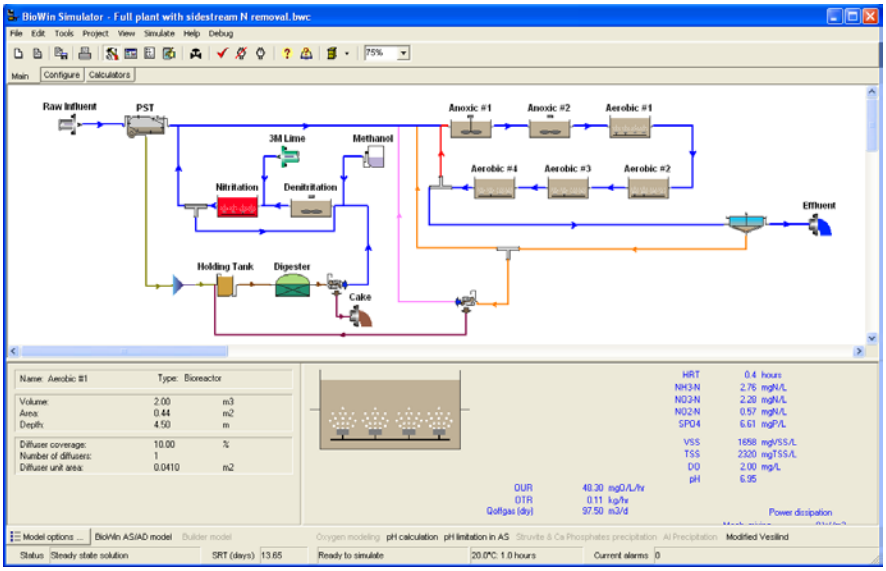
It's easy to use. The program has the look and feel of the many other Windows applications. When it is launched it comes up with the familiar interface and menu structure. Complex treatment plant schemes can be configured rapidly through "drag and drop" mouse actions. Functions are selected from the pull-down menus, using short cut keys, or by pointing the mouse and clicking on icons in the toolbar. The user can also access many of the Windows functions usually embedded in a

Windows application; for example, selecting and configuring the printer setup. Context-sensitive Help is built into BioWin to provide on-line assistance, particularly for new users.

Careful consideration has gone into the design of the package; for example, the hardware and software platforms, the object oriented software development system, the data structures, the user interface, and so on. A primary aim has been the production of a package structured to allow on-going development in years to come.

The BioWin simulator suite presently includes two modules:

- A **steady state module** for analyzing systems based on constant influent loading and/or flow weighted averages of time-varying inputs. This unit is also very useful for mass balancing over complex plant configurations.
- An **interactive dynamic simulator** where the user can operate and manipulate the treatment system "on the fly". This module is ideal for training and for analyzing system response when subjected to time-varying inputs or changes in operating strategy.



About The Manual

This manual is available to you in both printed and online format. The content of both formats is *identical* – the choice is offered merely for your convenience. For information on printing the manual, please see **How To Print the Manual**. For information about the online help system, please see **On-line Help**.

Layout

Material in this manual divides into several parts:

- **Chapter 1** (this chapter) introduces you to BioWin.
- **Chapter 2** gives a quick tour of BioWin features.

- **Chapter 3** provides an overview of the on-line help system available in BioWin and how it may be used to deliver information.
- **Chapter 4** contains information on BioWin's main simulation window.
- **Chapter 5** describes the different element types (i.e. unit processes) available in BioWin.
- **Chapter 6** introduces the BioWin album.
- **Chapter 7** examines the various methods available for creating charts and adding series to them.
- **Chapter 8** provides in-depth chart formatting procedures.
- **Chapter 9** provides in-depth series formatting procedures.
- **Chapter 10** looks at BioWin's powerful project management tools.
- **Chapter 11** gives an overview of the tools available for customizing BioWin to match your preferences.
- **Chapter 12** contains descriptions of useful BioWin interface tools and techniques.
- **Chapter 13** contains a set of tutorials and case studies designed as a training exercise in the application of BioWin.
- **Chapter 14** describes the various functional elements of the BioWin General Model.
- **Chapter 15** outlines the theory behind the three basic types of models BioWin uses in solid / liquid separation elements.
- **Chapter 16** details BioWin's approach to pH modeling.
- **Chapter 17** explains the gas-liquid mass transfer model used in BioWin
- **Chapter 18** provides a general description of the modeling of anaerobic processes in BioWin particularly as they would occur in anaerobic digestion processes.
- **Chapter 19** describes the biofilm model provided in BioWin. The model is implemented in the media bioreactor element and is calibrated to simulate MBBR and IFAS systems..
- **Chapter 20** describes the sidestream model provided in BioWin.

Typefaces and conventions used in this manual

The following typefaces and conventions will be used throughout this manual when describing various procedures and techniques in BioWin:

Typeface	Refers To
Example Text	Italic text refers to chapter titles.
Example Text	Bold text in Arial font is used when describing objects and controls in dialog boxes, as well as the dialog box names. When you see text in this typeface used in

	the description of a dialog, that text will be on the dialog box.

How To Print The Manual

The BioWin manual is shipped in the form of one complete Adobe PDF. The content of this document is identical to BioWin's online help – the help was built using the manual document as its source (Note that you also may print out individual topics from within the online help system). If you wish, you may install this document to your computer and print it out as you desire. However, if you wish to save disk space, you may choose to leave it on the CD, where it is located within the **MANUAL** directory.

Online Help

BioWin comes equipped with full featured, context-sensitive on-line help. The help features an expanding/collapsing table of contents, a full multi-level keyword index, and a full text search. The on-line help is built from the manual, so users will have a number of options available to them when in comes to accessing information.

The help format used in BioWin is known as **HTML** help. This format commonly is used in Microsoft Office applications, and will be familiar to most users. This format delivers the help system via a two-paned web browser interface that displays the help system topic structure in the left pane and the selected topic in the right pane. In this way, users can see where they are in the overall topic structure as they browse through the help system. For more information on this style of help, please see the chapter entitled "*Getting Help In BioWin*".

Limitations

BioWin is a very powerful analysis tool. The program has been evaluated against an extensive data set and has been demonstrated to provide accurate simulation results for a range of systems. Nevertheless, the user is cautioned that BioWin is merely a tool.

BioWin incorporates a number of models. These necessarily are not perfect and have limited ranges of applicability. It is the responsibility of the user to carefully assess results generated by the program.

New Developments in BioWin

What is New in Version 3

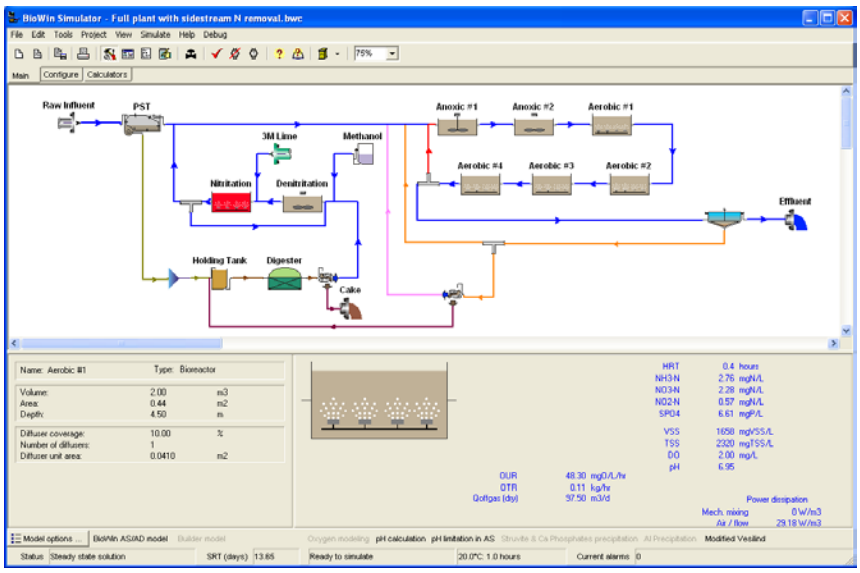
The latest version of BioWin provides a host of additions and improvements to enhance your wastewater treatment plant simulations. The main additions to Version 3 in terms of modeling capacity are:

- Biofilm model
- Two-step nitrification and denitrification
- Anammox bacteria

Incorporation of two-step nitrification and denitrification and the growth of Anammox bacteria enhance the capacity of BioWin for detailed modeling of the various sidestream (e.g. digester centrate) treatment systems that have been developed in recent years.

Note: A new Sidestream Reactor element can be included in configurations. This is mainly for convenience as it is easily distinguished from other activated sludge reactors on the drawing board. The model applied in a sidestream reactor is no different from the model used in other units. BioWin is based on a single integrated model for all biological and chemical reactions, and the same model is applied to any unit in a BioWin simulation. The only difference for a Sidestream Reactor is that the "seed" values selected by BioWin when a simulation starts differ from those for a standard activated sludge bioreactor.

Some of the additions, changes and upgrades in BioWin 3 are listed here. [A number of the features added to BioWin Version 2 are also listed as a reminder to users].



Model Additions and Enhancements

The additions to the **Full Plant Edition** of BioWin are significant advances in modeling of wastewater treatment plants. BioWin tracks more organic and inorganic components than any other simulator, and allows a complete mass balance for recycled sidestreams and supernatants. The integrated process model in BioWin works for any environmental condition, whether in aerated or unaerated activated sludge tanks, fermenters, sidestream reactors or digesters. This allows seamless integration of the processes in the whole plant.

A few Version 3 highlights

- Development of a sophisticated biofilm model. This has been implemented initially only for Media Bioreactors; that is, activated sludge suspended growth reactors that contain a free-floating carrier media for biofilm growth e.g. integrated fixed-film activated sludge (IFAS) and moving bed bioreactor (MBBR) systems. The Media Bioreactor module also can be used to configure reasonable representations for other biofilm systems such as trickling filters, biological aerated filters, and tertiary denitrification filters. For further details on the biofilm model review the "*Biofilm*" chapter.
- Modeling nitrification as a two-step process; that is, conversion of ammonia to nitrite (NO_2) – mediated by ammonia-oxidizing bacteria (AOBs), and conversion of nitrite to nitrate (NO_3) – mediated by nitrite-oxidizing bacteria (NOBs). For further details review the "*Sidestream*" chapter.
- Denitrification by heterotrophs [ordinary heterotrophic organisms (OHOs), phosphorus-accumulating organisms (PAOs), or methanol-utilizing heterotrophs (methylotrophs)] is modeled as a two-step process with conversion of nitrate to nitrite and then nitrogen gas.
- Modeling the growth of Anammox bacteria; autotrophic organisms that combine ammonia and nitrite to form nitrogen gas without the addition of organic substrate. For further details review the "*Sidestream*" chapter.

Version 2 highlights

- BioWin includes accurate pH modeling and pH dependence of biological and chemical processes.
- Integrated biological model for BNR activated sludge, fermenters, and anaerobic digesters.
- Alum or ferric dosing model for phosphorus precipitation and chemical sludge formation integrated with the biological model. pH dependent precipitation kinetics.
- Spontaneous struvite and hydroxyapatite precipitation processes.
- Detailed tracking of inert solids content, and N and P balances in biological and chemical sludge.
- Fast pH simulation using a unique model solution based on concentrations of strong acids and bases, the dissociation state of the phosphate, carbonate, ammonium and volatile organic acid system, chemical precipitation reactions, and stripping of components such as ammonia and carbon dioxide.
- Improved modeling of denitrification with methanol addition - methanol-specific biomass, adaptation, minimum anoxic SRT, and substrate specific anoxic yields.
- Magnesium and potassium uptake with biological phosphorus uptake. Release of stored magnesium in digesters.
- Integrated digester model describes acidogenic phase (prefermenters) and methanogenic phase, predicts gas flow, composition and pH.
- Surface aerators and brushes in addition to diffused aeration.
- Improved, intuitive, customizable interface. Key variables for each process (OUR, Nitrate Production Rate, Specific Denitrification Rate, etc.) readily accessible.
- Model Builder enables users to customize models or create their own - starting from scratch or by modifying existing models.
- The International Modeling Kit includes standard models such as the ASM series and the double-exponential settler model.

Usability Enhancements

A number of features have been added to BioWin to streamline ease of use.

Version 3 new features

- In ideal separation devices (point settlers, dewatering units, and ideal clarifiers) a schedule can be defined to specify time varying percentage removals.
- Oxygen transfer calculations are based on diffuser submergence (rather than tank depth).
- When setting up charts in the Album, the list of variables that can be plotted is very extensive. These have been grouped in separate category lists (state variables, combined variables, water chemistry

variables, commonly-plotted variables). This simplifies selecting variables to plot.

- Improved numerical solution techniques and more simulation method options.
- Expanded and enhanced COM interface – including read/write access to influent totals and fractions as well as project temperature. Read-only access to all liquid phase state variables and the gas phase states of anaerobic digesters.
- The facility to plot profiles of variable concentrations through layers in the biofilm in Media Bioreactors.
- Itineraries are now completely independent of the "constant" value.
- Additional alarms to alert the user to unusual conditions.
- Thread / process priority management for BioWin users.
- The ability to specify project start time from the project info dialog box.
- Improved grid snap methods.
- Oxygen transfer through the surface in anoxic zones.
- Improved "fly-by" panels.
- The facility to include / exclude media from media reactors.

Features added in Version 2

- State Point Analysis plot. From BioWin 2.2 the user can plot the state point analysis flux curve for ideal and model secondary settlers as well as for SBR elements (some restrictions apply).
- Chart annotation tools.
- Multiple SRT calculation scenarios.
- Plot SRT against time during dynamic simulations.
- Automatically save dynamic simulations as they progress.
- Parameters may be "unmonitored".
- Floating / docking toolbars.
- Improved numerical procedures for steady state solver.
- Locate a user-defined concentration within a model settling tank depth/concentration profile (e.g. use this to find where you think the sludge blanket is).
- Display full or abbreviated names for state variables and other parameters.
- The new Chart Master allows you to cascade multiple changes through selected charts in the BioWin Album.
- Further improvements to powerful Report to Word™ option.

Opening a File from a Previous Version

When you open a file created in an earlier version of BioWin, you will be presented with a dialog box that contains "conversion notes". These warnings reflect changes that have been made to the underlying model structure of BioWin.

The content of the conversion notes depends on the whether the file being loaded was created in Version 2 or in an earlier version of BioWin. The most important aspects to recognize when loading a file from a previous version are:

- **Review inputs:** BioWin Version 3 has several additional state variables. You should review your input elements to check that you have appropriate values for the new state variables and/or wastewater characteristic fractions.
- **Check model parameters:** The new version incorporates additions and some modifications to the activated sludge and anaerobic digestion models. It is imperative that all model parameters (kinetic, stoichiometric, switching function parameters, etc.) should be checked carefully when loading a file created in an earlier version.
- **Re-run the simulation:** When loading a file created in an earlier version of BioWin, you must re-run any simulations to get meaningful results.

Conversion Notes

Conversion notes are displayed when you open a file created in an earlier version of BioWin. These are intended to prompt you to make changes in the BioWin file that will make it compatible with the latest version. It is imperative that all model parameters (kinetic, stoichiometric, switching function parameters, etc.) should be checked carefully. The following sections provide brief explanations for the different messages that appear, depending on the file version being loaded.

All Alkalinities Will Be Converted

Alkalinity is no longer a state variable in BioWin. In influent streams you can still enter alkalinity (and pH) and BioWin will calculate the required dissolved CO2, anions and cations. BioWin still calculates the alkalinity, but now it is determined from the equilibrium chemistry of the stream, rather than by a pseudo state variable.

Additional State Variables Included - Review Inputs

BioWin now has several additional state variables. You should review your input elements to check that you have appropriate values for the new state variables and/or wastewater characteristic fractions, and that an appropriate pH is specified.

Additional State Variables Included AND Parameters Should Be Changed And New Parameters Added

The new version incorporates additions and some modifications to the activated sludge and anaerobic digestion models. In the case of model additions (e.g. Anammox bacteria), BioWin will assign default values to all model parameters for the new model components. In the case of model modifications (e.g. two-step nitrification) parameter values from the earlier version will be mapped to the appropriate parameter in Version 3, and default values will be assigned to new model parameters. Also, some small changes have been made to certain default parameter values. It is imperative that all model parameters (kinetic, stoichiometric, switching function parameters, etc.) should be checked carefully when loading a file created in an earlier version.

Unable to convert Alkalinity

When loading an old file BioWin attempts to match the old alkalinity, by adding acid or base. If it is unable to achieve a match this note occurs. If you see this note you should check your influent streams and pH's.

Alkalinity Is No Longer A State Variable - Equation Discarded

Alkalinity is no longer a BioWin state variable (it is still reported, but now it is determined from the equilibrium chemistry of the stream, rather than by a pseudo state variable . This message occurs because you have used Alkalinity in a Model Builder model (previously called the **Grey Box**), but since it is no longer available as a state variable you will need to adjust your model to account for this.

Enter Head Space Volume For Digesters

The gas phase of the anaerobic digester element is now modeled, and you must specify the volume that BioWin should use for these calculations.

Methane Production Now Reported As A Fraction

Methane is now reported as a fraction – in earlier versions it was reported as a Methane production rate. You will need to adjust you chart scales to see the new value (which should always be below 1).

Hydrogen Production Now Reported As A Fraction

Hydrogen is now reported as a fraction – in earlier versions it was reported as a Hydrogen production rate. You will need to adjust you chart scales to see the new value (which should always be below 1).

OUR And Denitrification Rate No Longer Reported For Anaerobic Digesters

OUR and denitrification rate are no longer available for anaerobic digesters (OUR should generally be essentially zero anyway).

Variable XXX No Longer Available - Check Your Graphs

The variable "XXX" is no longer available – but BioWin has detected that you were monitoring it. You will need to check your graphs as any series based on this will now be "static" showing only the values from the last simulation in a the previous version. Generally if a variable has been removed from BioWin it has been replaced by a similar, usually more informative one.

Aeration Parameters Can Now Also Be Global

Aeration parameters can be globally specified now. Old files are loaded with local aeration parameters so that any settings like Alpha, beta etc. are preserved. In certain cases you may wish to have "local" aeration parameters (for example in a pure oxygen system where you need to change the oxygen content of the gas). You can do this by selecting the **Local parameters** check box on the **operation** tab.

New program version - resimulate to get meaningful results.

The file loaded was created in an earlier version of BioWin. You must re-run any simulations to get meaningful results.

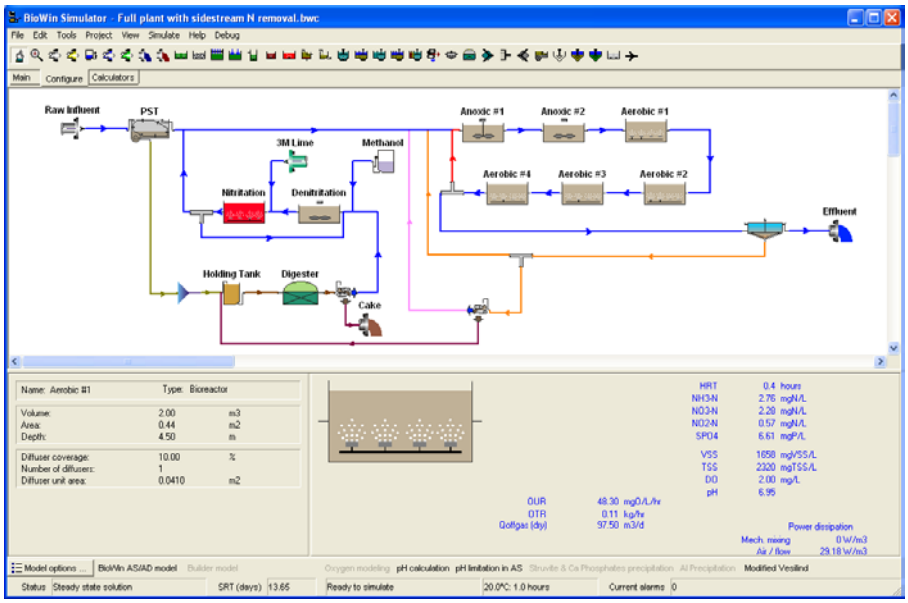
Quick Feature Tour

Quick Feature Tour Overview

This chapter highlights some of the features available in the latest version of BioWin. These are demonstrated using the "An Example" configuration installed in the **Data** directory. The purpose of this chapter is to provide a brief introduction.

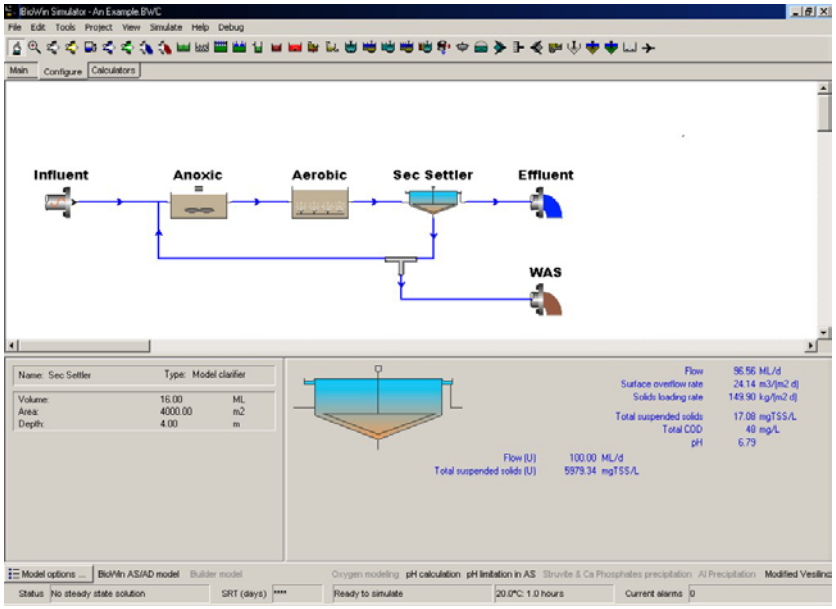
For specific examples on using BioWin, please see "*BioWin Tutorials*". You can find more BioWin examples as follows:

- Select **File|Open** and browse to the **Examples** directory. These systems are discussed in the Help section on BioWin Tutorials and Examples.
- On the BioWin main window toolbar, at the end on the right, click on the arrow next to the icon that looks like a filing cabinet. This brings down a list of pre-configured BioWin process files for a range of system configurations.



The Interface

The example system shown below is a simple two-reactor activated sludge configuration.



A simple two-reactor activated sludge configuration

The BioWin main simulator window interface consists of:

- Menus
- Toolbars
- Drawing Board
- Summary Panes
- Status Bar

Setting up such a system is easy to do – it's a matter of minutes. Buttons on the toolbar at the top of the main simulator window represent the various unit processes available in BioWin. Simply click on a button, move your mouse cursor over the area on the drawing board where you want to place an element, and click the mouse button.

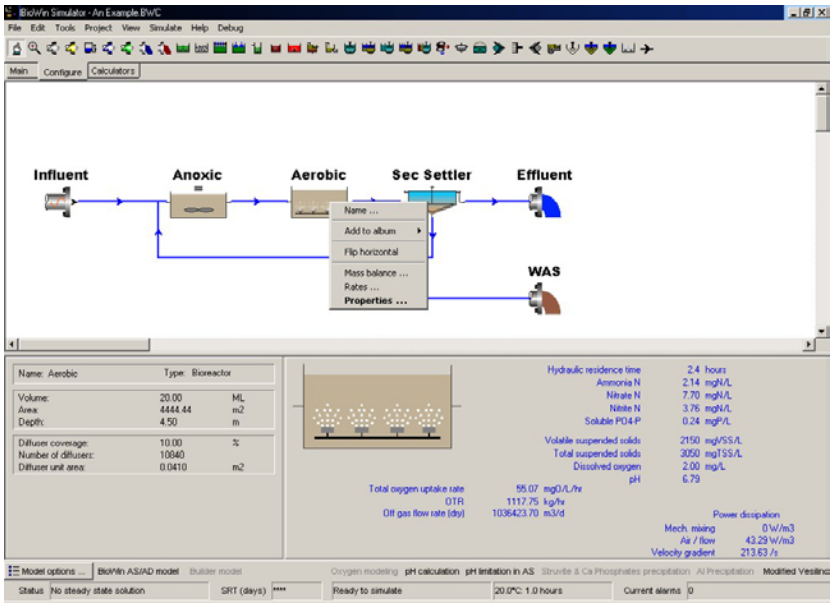
Most types of wastewater treatment systems can be configured in BioWin using the many process modules. These include:

- A range of activated sludge bioreactor modules – suspended growth reactors (diffused air or surface aeration), various SBRs, media reactors for IFAS and MBBR systems, variable volume reactors.
- Anaerobic and aerobic digesters.
- Various settling tank modules – primary, ideal and 1-D model settlers.
- Different input elements – wastewater influent (COD- or BOD-based), user-defined (state variable concentrations), metal addition for chemical phosphorus precipitation (ferric or alum), methanol for denitrification.

- Other process modules – holding tanks, equalization tanks, dewatering units, flow splitters and combiners.

Note: A new Sidestream Reactor element can be included in configurations. This is mainly for convenience as it is easily distinguished from other activated sludge reactors on the drawing board. The model applied in a sidestream reactor is no different from the model used in other units. BioWin is based on a single integrated model for all biological and chemical reactions, and the same model is applied to any unit in a BioWin simulation. The only difference for a Sidestream Reactor is that the "seed" values selected by BioWin when a simulation starts differ from those for a standard activated sludge bioreactor.

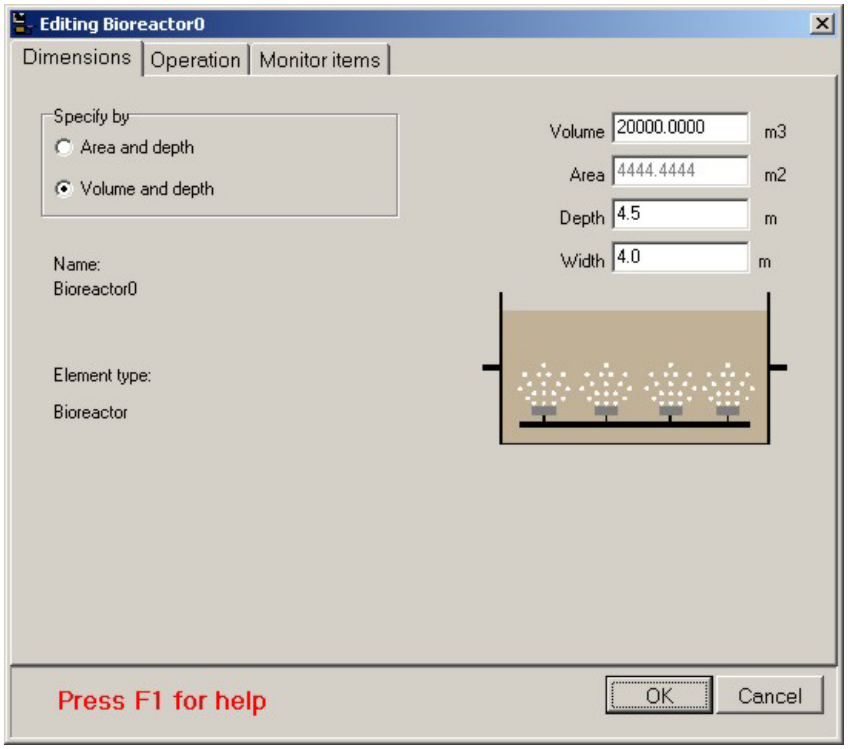
A quick way to gain access to local menus which contain commands specific to a particular object is through the use of the right mouse button. For example, if you point to the influent element and right-click, you will get a local menu as shown below.



Using the right mouse button gives access to local menus

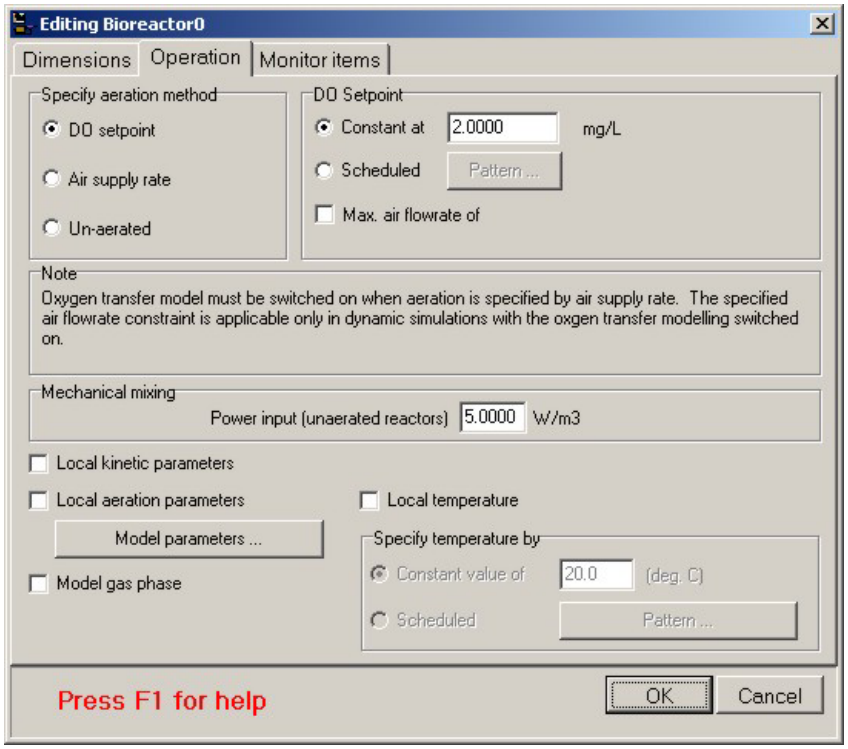
Element Information

Double-clicking on a drawing board icon for an element in a configuration gives access to all pertinent information for that element. For example, double-clicking on a bioreactor element allows access to physical and operational data, as well as the facility to set up data monitoring.



Dialog box allows access to all bioreactor information

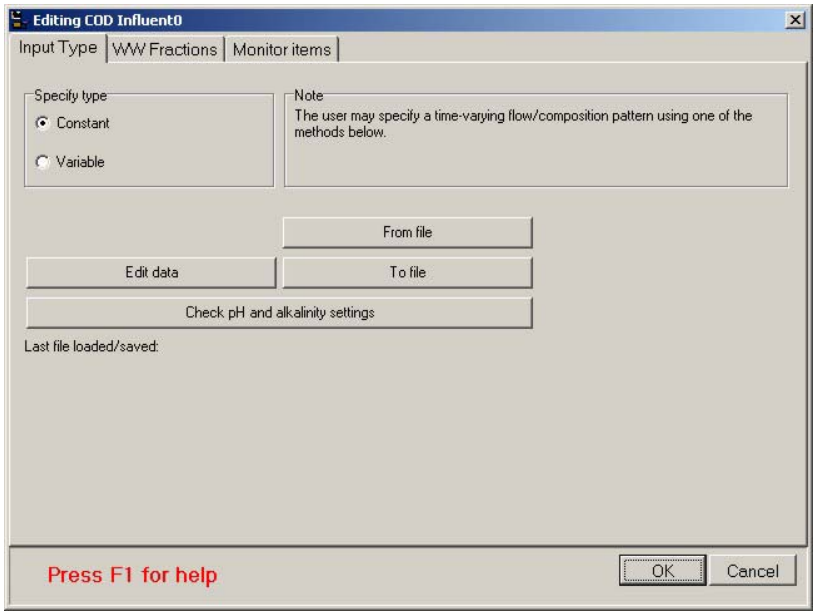
Once you've double-clicked on an element icon to gain access to this information, it's just a matter of clicking on the tab you are interested in. For example, clicking on the **Operation** tab will allow you to change the bioreactor operating parameters shown below.



The bioreactor operation tab

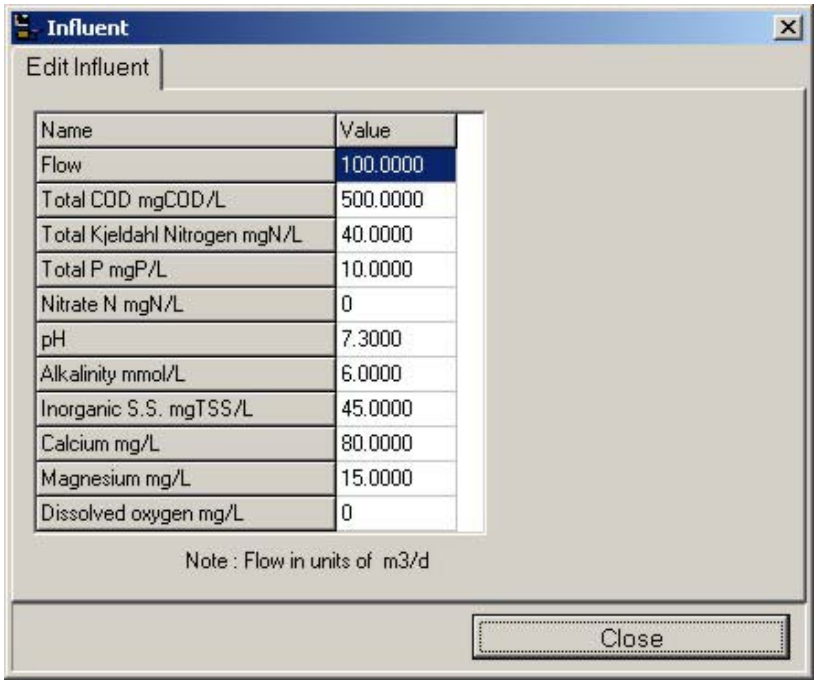
Influent Data

Setting up influent data is a quick and easy process. If you double-click on an influent element drawing board icon, you will see the following dialog box.



Access the influent properties to set up influent data

Clicking the **Edit data** button as shown will open the **Influent itinerary editor**, as shown below.



The variable influent itinerary editor

The **Influent itinerary editor** provides a spreadsheet-like interface for entering data. BioWin even offers several different strategies for filling in blanks in your data! It is very easy to import data into the itinerary editor from files or to copy it in from a spreadsheet – in fact, the data in the example shown above were pasted in from Microsoft Excel™!

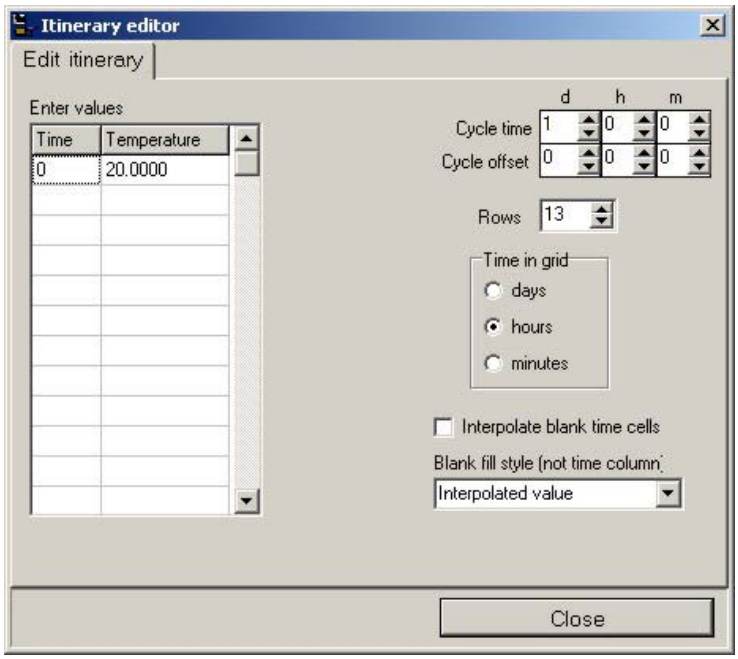
Running A Simulation

Steady state balance tool

Dynamic simulation tool

Steady state and dynamic simulations are run from the main simulator window. Once you have taken a few minutes to specify information for the various elements in your configuration, commencing a simulation is simply a matter of clicking the appropriate button on the toolbar!

BioWin uses a powerful itinerary that allows the user to schedule many different operating conditions such as dissolved oxygen setpoints, air flowrates, and temperature. For example, suppose that you were simulating varying temperature conditions. With BioWin, you easily can set up a temperature schedule using the dialog box shown below.



Scheduling operating conditions is not a problem!

Help and Manual

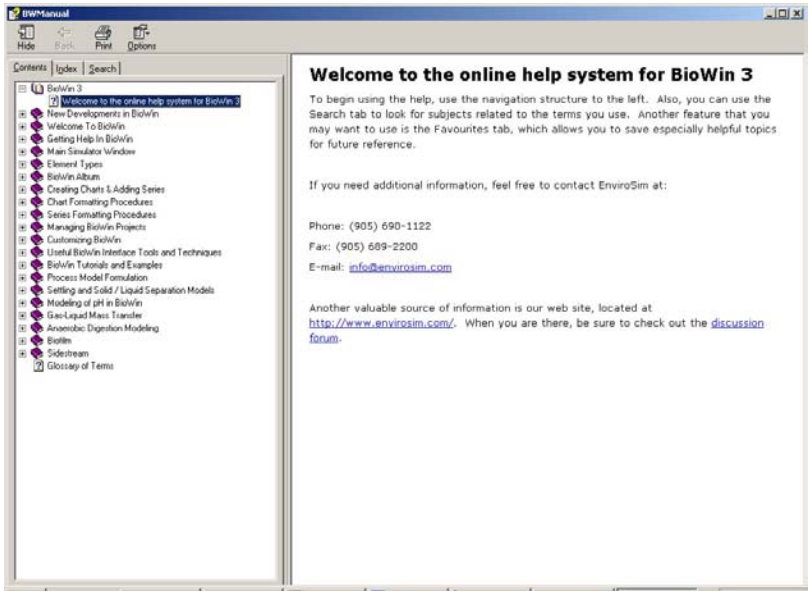
BioWin comes with an extensive manual which is shipped as a series of Microsoft Word™ documents each consisting of a chapter so you can easily print out the sections of specific interest.

Help Contents & Index

Help on using help

You may find this unnecessary as the contents of this manual are available via BioWin's online help. You can access this help system via the toolbar help buttons. Another useful feature that makes BioWin easy to learn is context-sensitive help. To get help that is relevant to a particular dialog box you are working in, simply hit the **F1** key and BioWin will access related topics from the help system and display them to you.

A screen shot of the help system is shown below.



A BioWin help window (Contents tab showing)

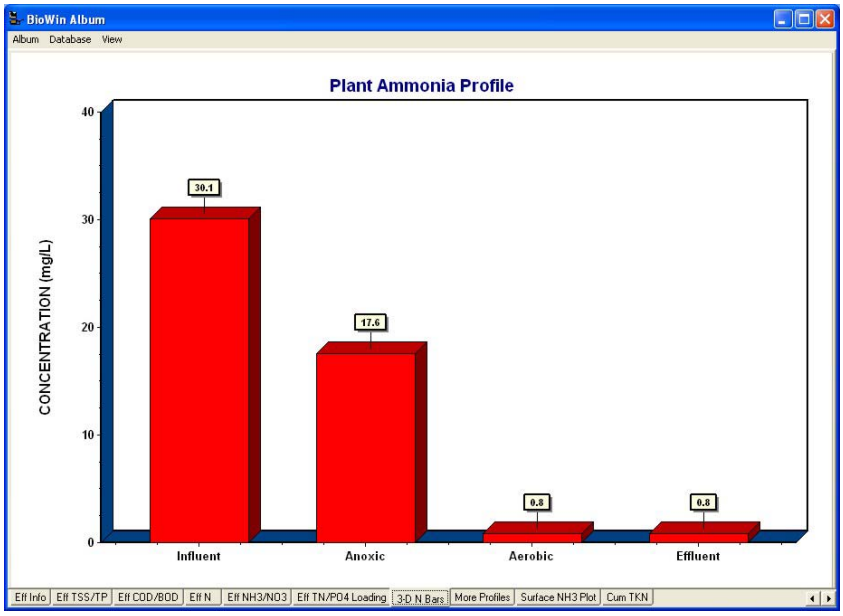
Viewing Simulation Results

The BioWin Album provides a fully integrated means to display simulation results. Using the album you can view data in the following formats:

- Tables
- Element-specific information displays
- Charts

 *Opens the album*

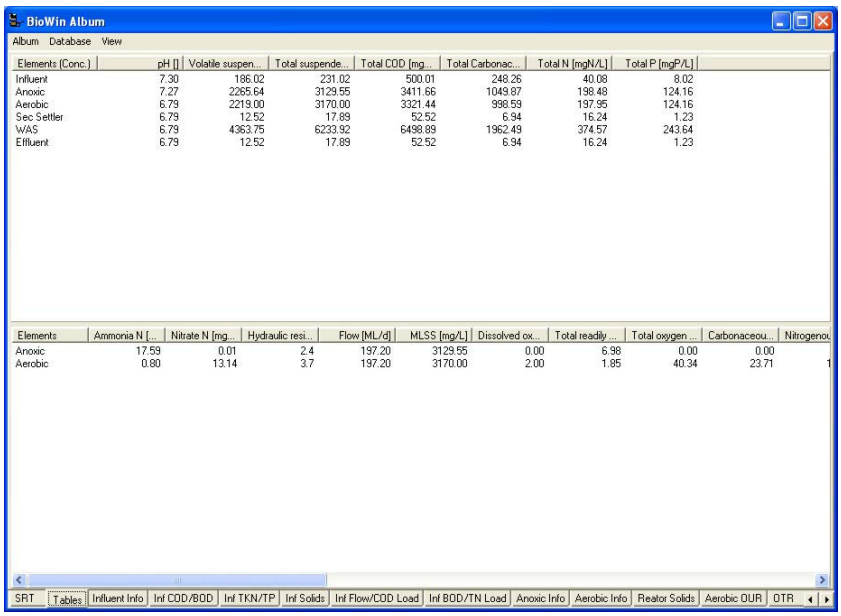
Activating the album is as simple as clicking the appropriate button on the main window toolbar. The album consists of a series of tabbed pages which may contain any or a combination of the above data display formats. Shown below is the album with the active page displaying a chart.



The album interface

Tables

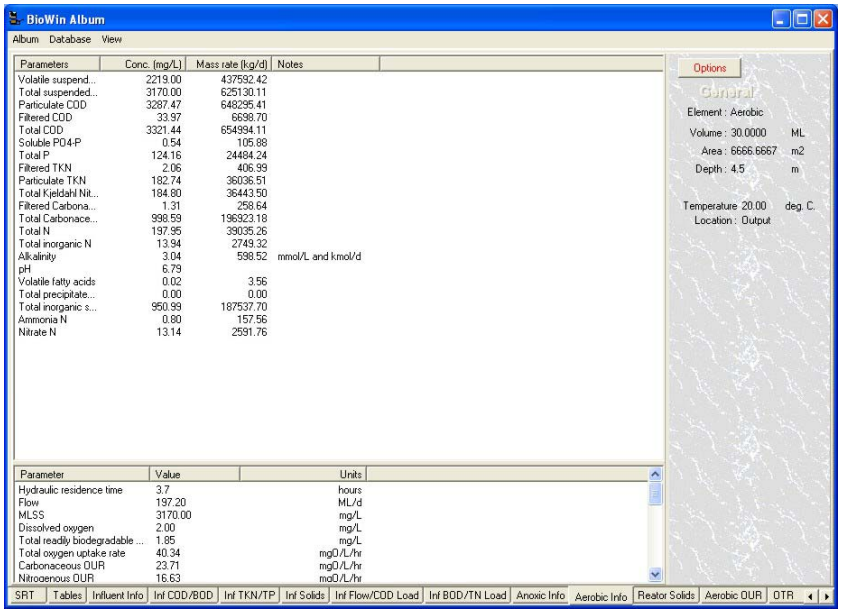
Here is an example of an album page containing two tables.



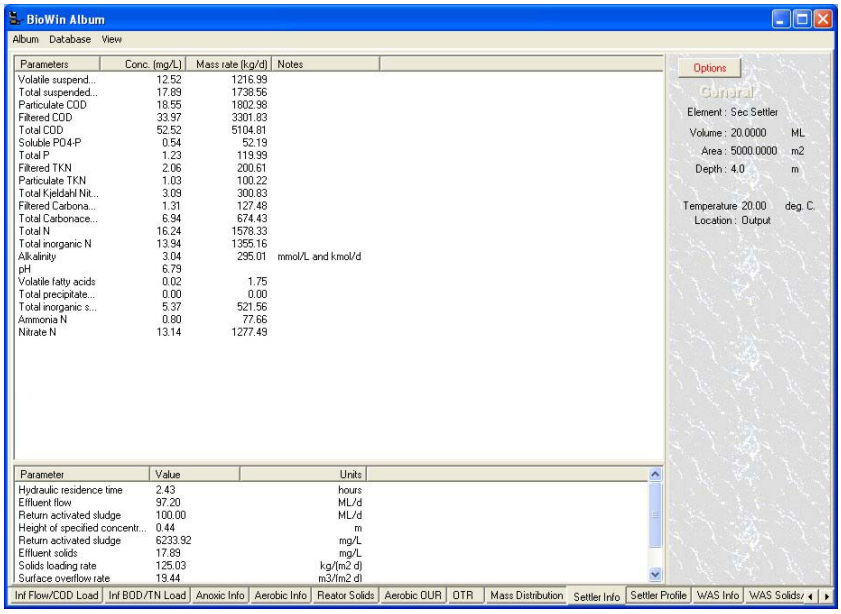
An album page containing two tables

Element-Specific Information

Here are two examples of element-specific information displays; one for a bioreactor element and one for a settling tank element.



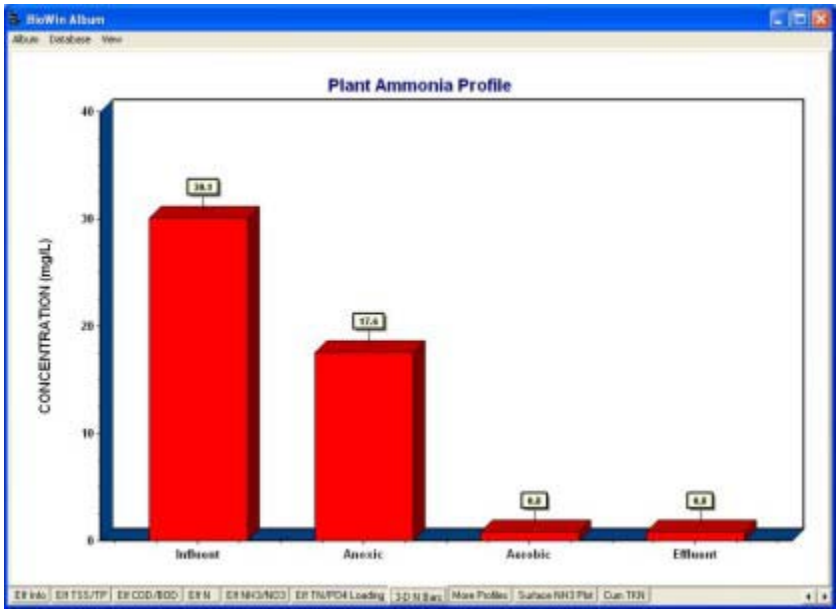
A bioreactor element-specific information display



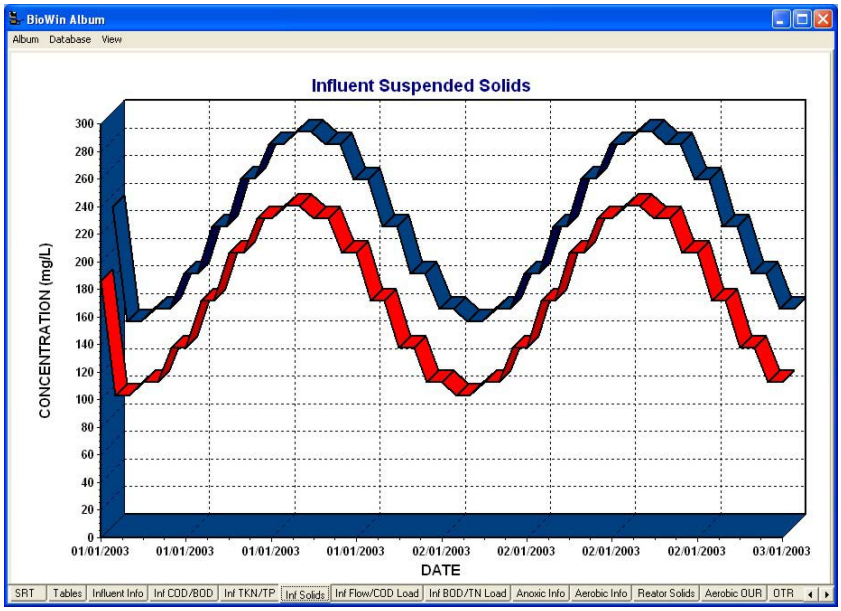
A settling tank element-specific information display

Charts

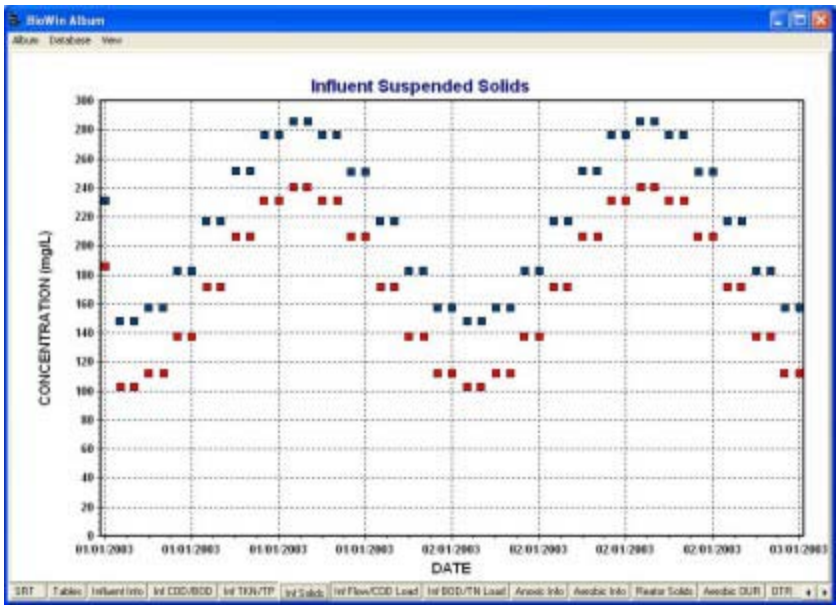
BioWin offers a wide variety of charting options. Here are some examples.



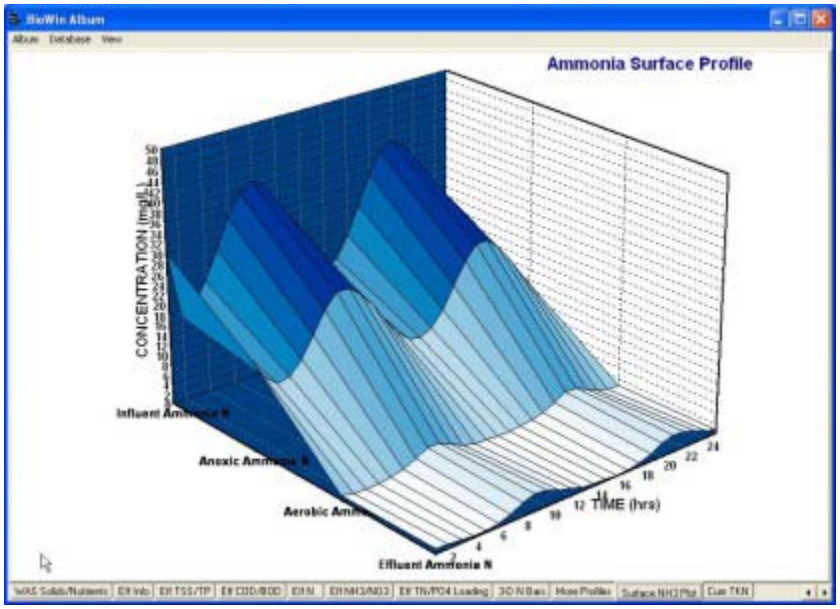
A bar chart



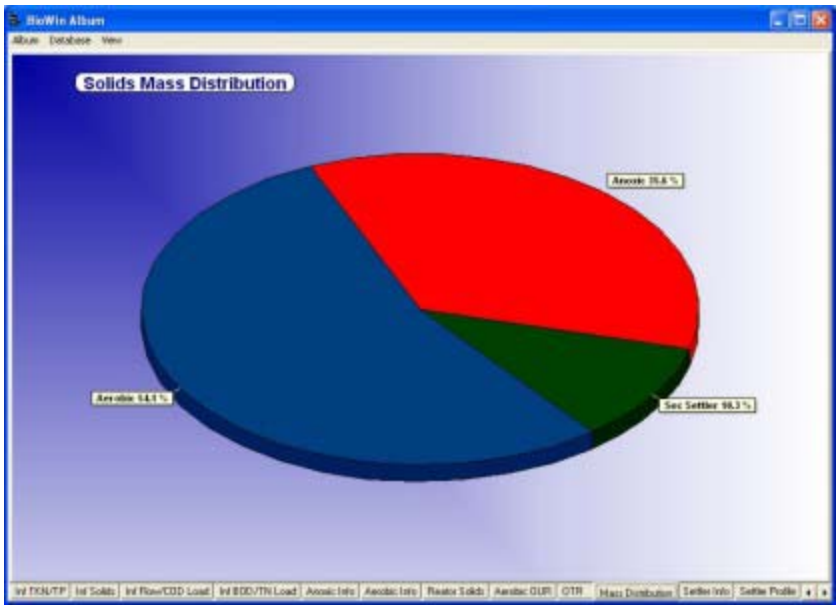
A three-dimensional line plot



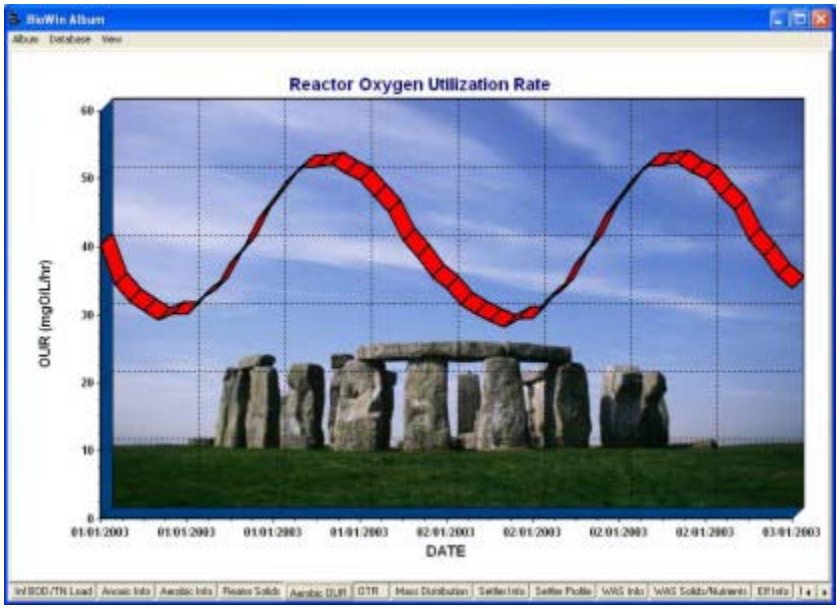
A two-dimensional point plot



A surface plot



A pie chart



Place pictures (or your company's logo!) in the chart background for presentations

Creating Reports

Printing Reports

BioWin incorporates a powerful automatic report generation feature. With a single click, BioWin can generate a detailed printed report.

Customizing Reports

The type of information that appears in the report is completely customizable. The general information that can be included in a report may be:

- Project information (user name, plant name, project name, etc.);
- A picture of the project flowsheet;
- Global model parameter values;
- Global temperature setting;
- Album pages (charts, tables, etc.);
- The SRT for the system (if one is available);
- Any notes that have been entered in the BioWin Notes editor;

The reporting can be customized to include element-specific information on an element-type basis. Users can choose whether or not they want to include information for element types (e.g. Bioreactor) in the report. The type of information included in the report for each type of element can be different and may include:

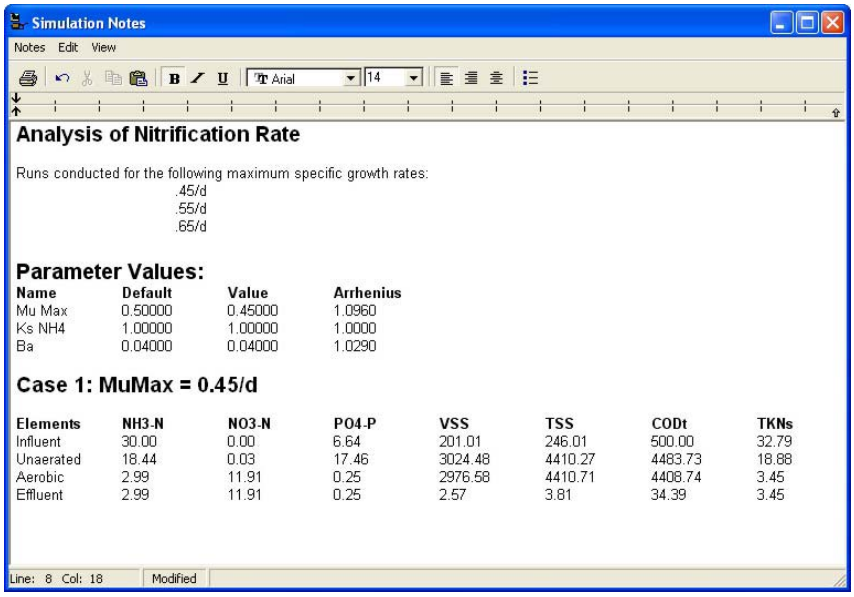
- Physical data (volume, area, depth, # of diffusers, etc.);
- Operating data (average or flow-weighted average);
- Local settling parameters (if available);
- Local biological model parameters (if available);
- Aeration parameters (if available);

Generating Reports in Microsoft Word™

If you prefer an electronic version, BioWin also can generate your report as a Microsoft Word™ document. Like the printed report, the information contained in the electronic version is completely customizable and may include many different forms. Once BioWin has generated the document for you, you may use it as a basis for an engineering report, or cut and paste its contents into another document.

Related functionality includes the ability to print out all or ranges of the album pages. You can also set the number of album pages per printed page.

Additionally, BioWin has its own internal **Notes** editor (shown below) to help keep track of project details.



BioWin's internal simulation notes editor

Exporting Results to a Word Processor or Spreadsheet

It also is very easy to get results from BioWin into your word processor or spreadsheet. Charts, tables, system configuration layouts, etc. can be copied and pasted from BioWin to your reports. Tables can be exported as tabbed text and then quickly converted to tables, such as the one below which is a section of a Word™ document.

Table 1

Elements	pH []	Volatile suspended solids [mgVSS/L]	Total suspended solids [mgTSS/L]	Total COD [mg/L]	Total Carbonaceous BOD [mg/L]	Total N [mgN/L]	Total P [mgP/L]
Influent	7.30	186.02	231.02	500.01	248.26	40.08	8.02
Anoxic	7.27	2265.64	3129.55	3411.66	1049.87	198.48	124.16
Aerobic	6.79	2219.00	3170.00	3321.44	998.59	197.95	124.16
Sec Settler	6.79	12.52	17.89	52.52	6.94	16.24	1.23
WAS	6.79	4363.75	6233.92	6498.89	1962.49	374.57	243.64
Effluent	6.79	12.52	17.89	52.52	6.94	16.24	1.23

A BioWin table exported to a word processing application

Customizing

There are a variety of features that can be customized in BioWin. These are outlined briefly below.

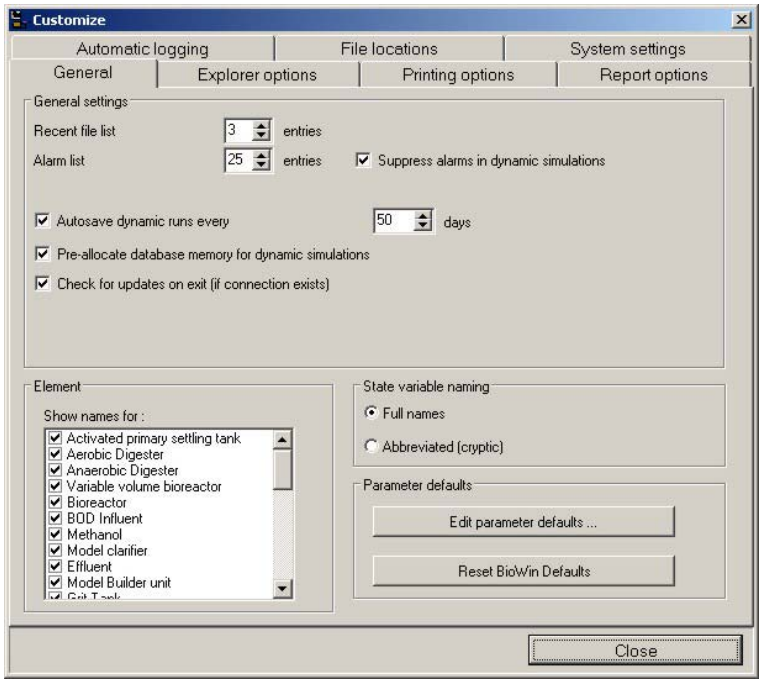
Customizing Environment Settings

BioWin offers users the ability to customize a number of environment settings to suit their needs. For example, some of the customizable features include:

- Printing options

- Report Options
- Automatic Logging
- File Locations
- Explorer Options
- System Settings

Access to the customizable features is managed through a central location, shown below.



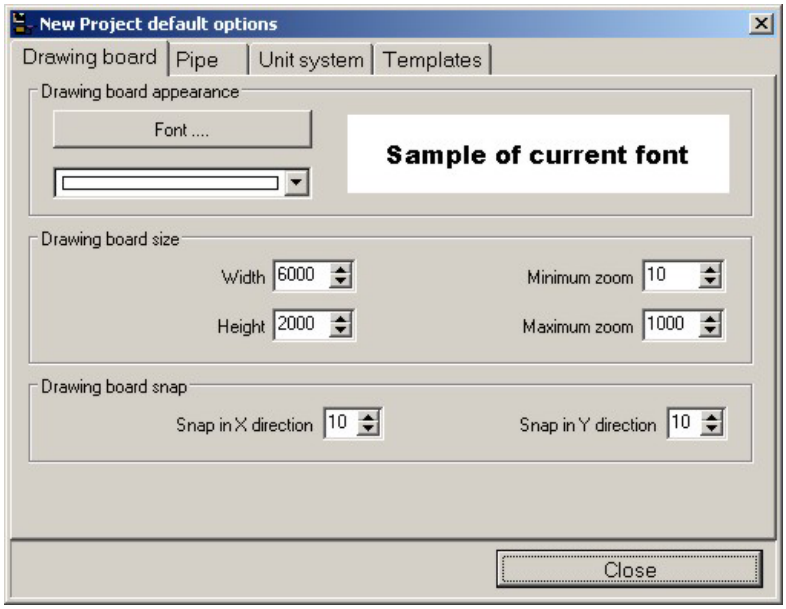
All customizable environment settings accessed through one dialog box

Customizing Project Settings

BioWin offers users the ability to customize a number of new project settings to suit their needs. For example, some of the customizable features include:

- Drawing board appearance
- Pipe Settings
- Unit System Settings
- Template Settings for the Album

Access to the customizable features is managed through a central location, shown below.



All customizable new project settings accessed through one dialog box

Customizing Charts

Finally, you can customize how BioWin generates new charts using the **Chart Master** and chart templates as shown below.

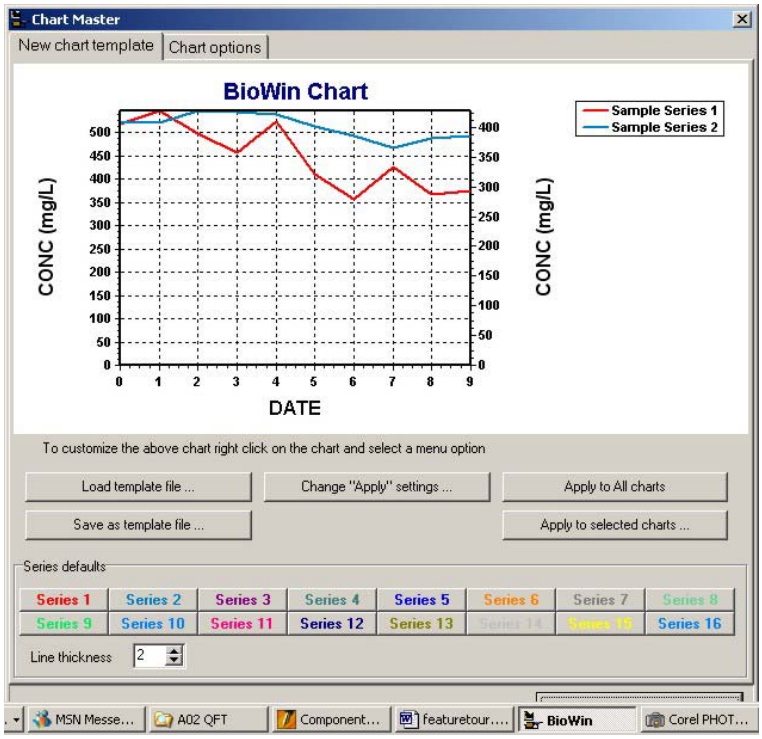


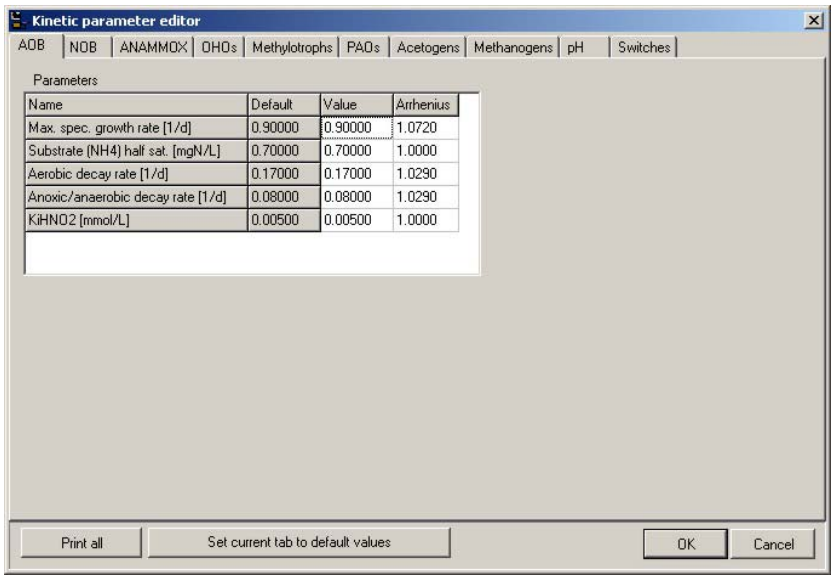
Chart Master

Model Information

BioWin is not only a slick simulator package. The user has ready access to detailed model features for the many operations. Model parameters may be accessed conveniently from a single **Model parameter editor**, shown below.

BioWin offers many utilities to facilitate process analysis. These include:

- Adjusting kinetic parameters and temperature in individual units;
- Simulation of biological activity in secondary clarifiers;
- Scheduling of many different operating parameters such as temperature, dissolved oxygen setpoint, air flowrate, and flow routing/splitting.



Model parameters may be changed through a single editor

Getting Help In BioWin

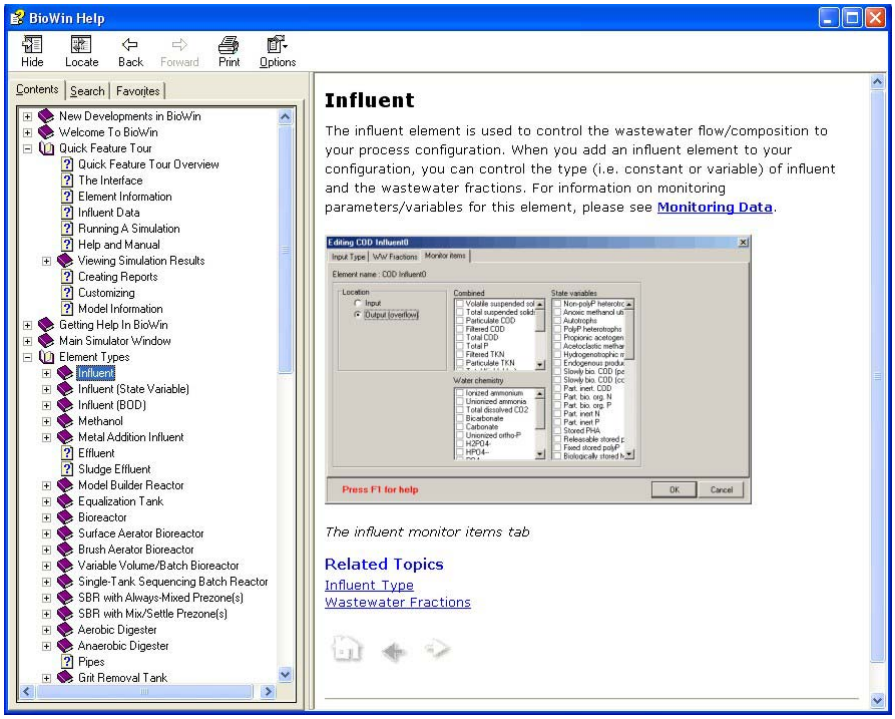
Using The BioWin Help System

BioWin's help system may be accessed with one of two methods:

1. The main simulator window menu command **Help|Contents and index**;
2. Clicking the **Help Contents and Index** button on the main toolbar of the main simulator window.

Help Contents & Index

When the help system is opened, you will see a two-paned window with the right pane showing the contents of the currently selected topic (or a default start topic if one is not selected) and the left pane showing the **Contents**, **Search,** and **Favorites** tab, depending on which tab was active when the help system was last exited. Note that the relative size of the panes can be changed by dragging the pane dividing bar. The two-paned window is shown below.



The Windows 98/HTML Help two-paned window

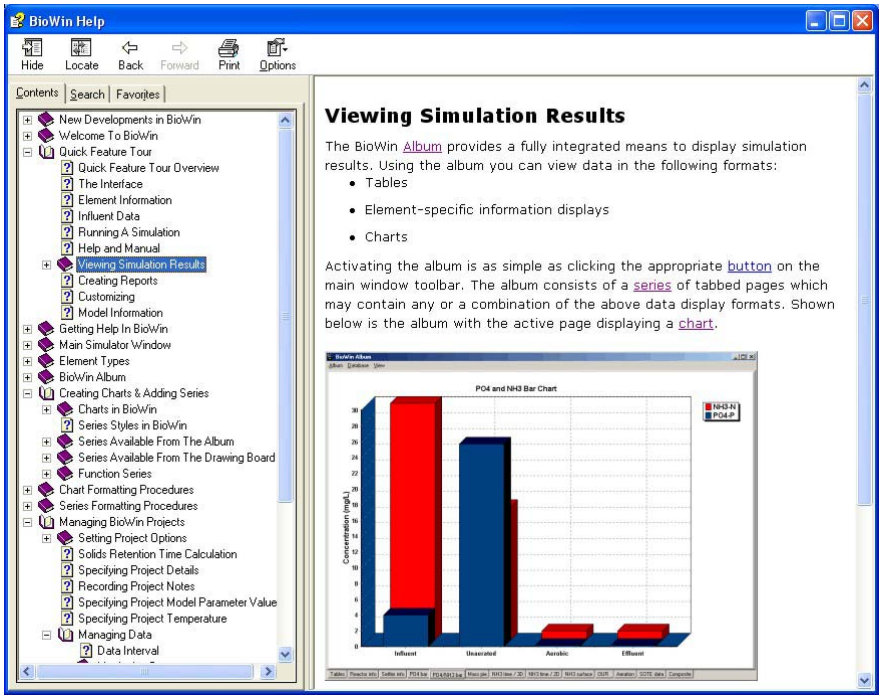
The buttons at the top of the two paned window have the following functions:

Button	Function
Hide / Show	Used to toggle the left pane between hidden and visible states.
Locate	Clicking this button will display the contents tab in the left pane and highlight the topic that you currently are viewing. This button is useful for locating related topics when using the Index or Search tab.
Back	Moves one topic back in your *browse history*. Takes you to the last topic that you viewed.
Forward	Moves one topic forward in your *browse history*. Takes you to the previous topic you viewed.
Print	A dialog box opens that allows you to choose whether you want the current topic or the current topic and all its sub-topics to be printed.
Options	Sets display options for your help system – it is recommended that these options be left set to their defaults.

Note that at the bottom of each topic there are two arrows. These are the **Browse Sequence** arrows, which may be used to move through the help system topic hierarchy one topic at a time.

Help Contents Tab

One advantage of the BioWin help is that when the **Contents** tab is selected, it is possible to view topics and the help system outline structure simultaneously, as shown in the picture below.

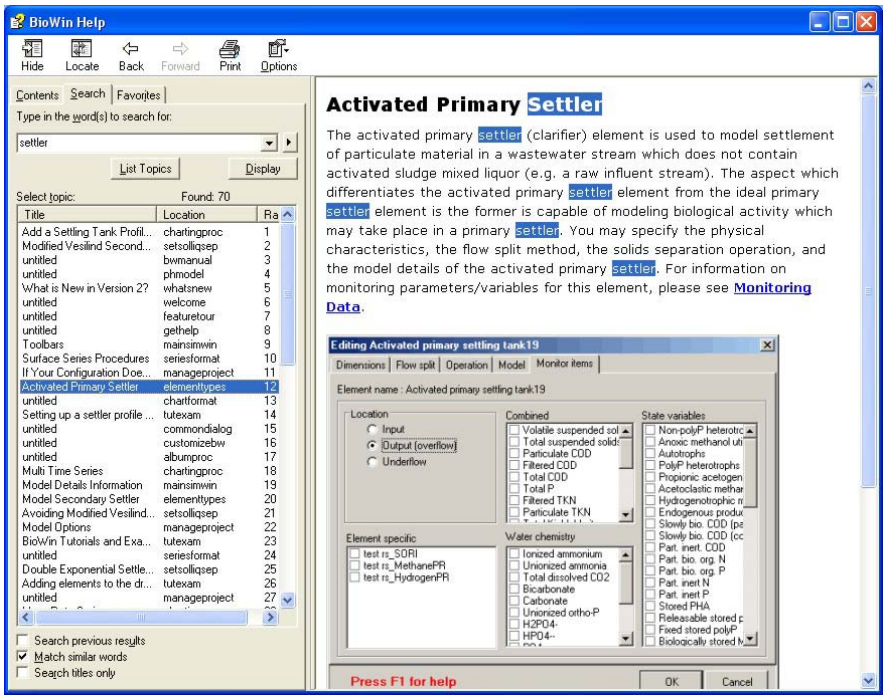


The Help Contents Tab

You can expand and collapse levels of the help system outline by clicking on the book icons or titles of the levels. If there is text associated with a level, it will be displayed in the right pane. When you locate the topic you wish to view, click on it and the topic contents will be displayed in the right pane.

Help Search Tab

When the Windows 98/HTML style help **Search** tab is selected, it is possible to view your search results and topics simultaneously, as shown in the picture below.

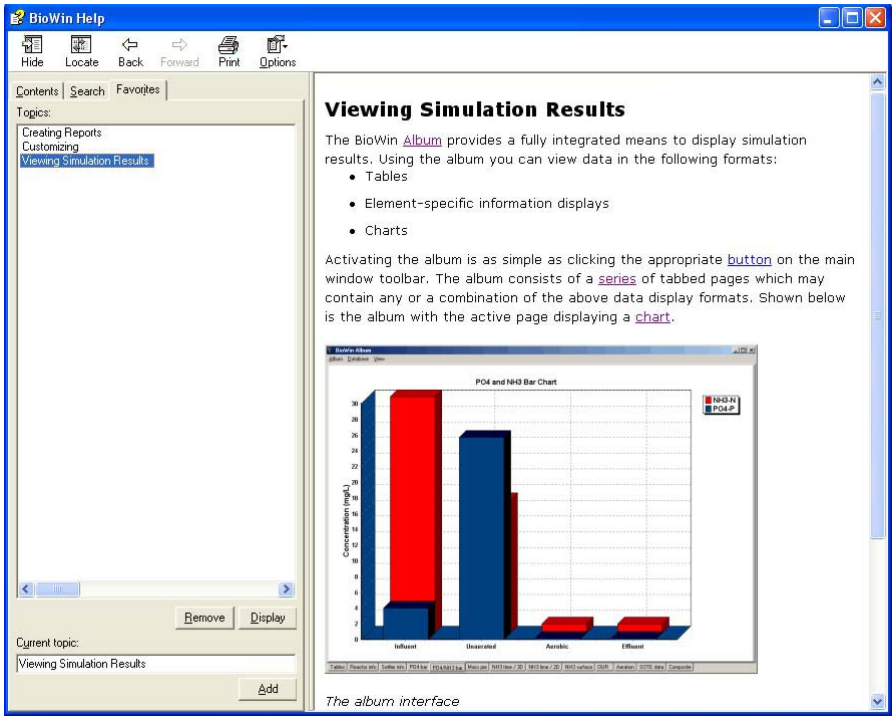


The Help Search tab

To use the search utility, type in the keyword(s) or phrase that you are searching for, and click the **List Topics** button. This will display the list of topics containing the keyword(s) or phrase that you searched for. To view a topic, you can either double-click on the topic title or click topic title and then click the **Display** button, and the topic contents with your search term(s) highlighted will be displayed in the right pane.

Help Favorites Tab

Another feature of the BioWin help is the **Favorites** tab, shown below.



The Help Favorites tab

Once you have found a topic using the **Contents** or **Search** tabs, you can add it to a list of favorites so that you can easily access it repeatedly. To do this,

1. Locate the topic that you want to add using the **Contents** or **Search** tab.
2. Click the **Favorites** tab.
3. The topic you currently are viewing will be displayed in the right pane, and at the bottom of the left pane the topic title will be displayed in the **Current topic** box.
4. To add the current topic to you list of favorites, click the **Add** button.
5. The topic will be added to the **Topics** list in the left pane.
6. To view other topics in the **Topics** list, double-click the topic, or click the topic then click the **Display** button.
7. To remove a topic from the list, click the topic title and then click the **Remove** button.

Popups and Jumps

In BioWin's help topics, you may encounter text that is blue and underlined. This means that the text represents a popup or hyperlink jump. If the text represents a popup, then clicking on the text will open a small "popup" window. Popups usually are used for definitions, small pictures, etc. When you are done viewing the contents of the popup, simply position your mouse cursor outside of the popup window and click once, or press any key on your keyboard to close it.

If the text represents a hyperlink jump, clicking on that text will send you to another location in the help file – usually a topic that contains material relevant to the text that denoted the jump. Note that when you do this, the help engine tracks your

position in the help system outline if you have the **Contents** tab selected so you know your location in the help system hierarchy. To return to the topic that you jumped from, click the **Back** button.

Context Sensitive Help In BioWin

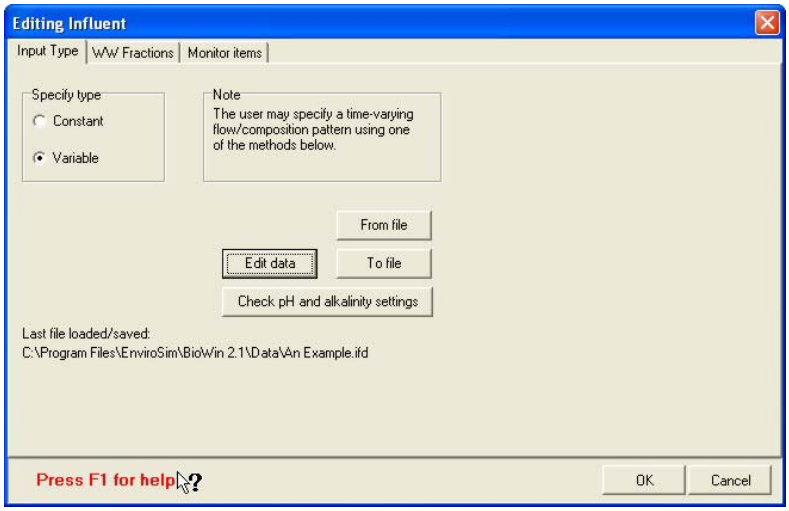
Context sensitive help is an advance in help; it is a system that makes it easier for users to find help immediately on topics that are relevant to the operations they are performing in BioWin.

When context sensitive help is called, the help system is opened with a topic that is related to the dialog box or menu command that you want help on. From this topic it is easy to navigate to other related topics.

There are a number of ways that the utility of context sensitive help may be employed:

Context Sensitive Help on a Dialog Box

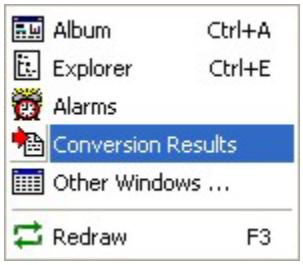
To get context sensitive help on a dialog box, press the **F1** key on your keyboard. Certain dialog boxes have red text at the bottom to remind you of this functionality. It also is possible to click on this text to invoke the help – the mouse cursor will display a question mark when you fly over this text, as shown below:



Dialog box with clickable context sensitivity reminder (notice the mouse cursor)

Context Sensitive Help on a Window Menu Command

To get context sensitive help on a window menu command, hold your mouse cursor over that command so that it is highlighted in blue as shown below and press the **F1** key on your keyboard.



A highlighted menu command

Context Sensitive Help on a Charting Dialog Box

Additional context sensitive help may be obtained for charting dialog boxes by clicking the **?** in the upper right hand corner and then clicking on a dialog box control.

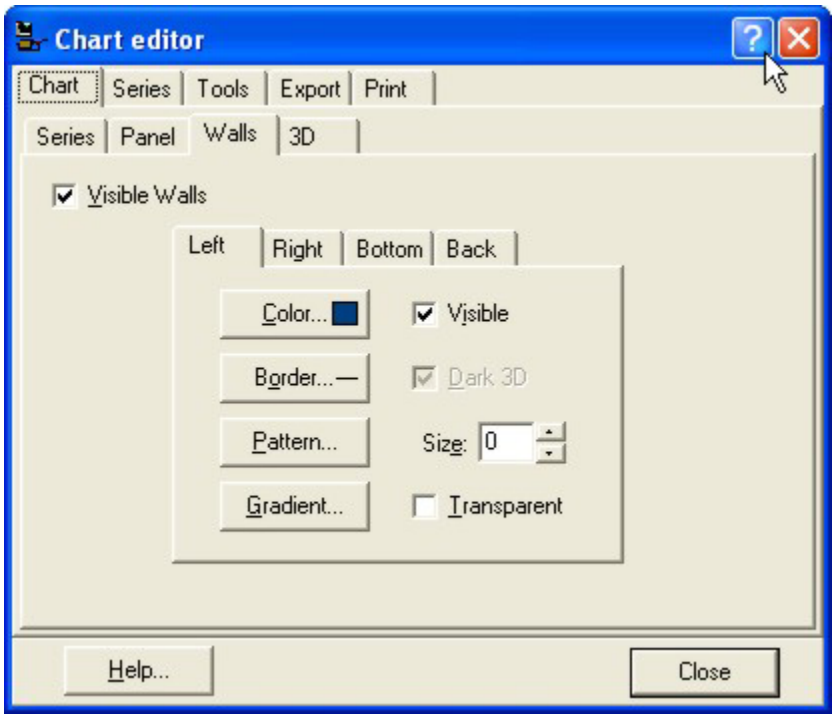


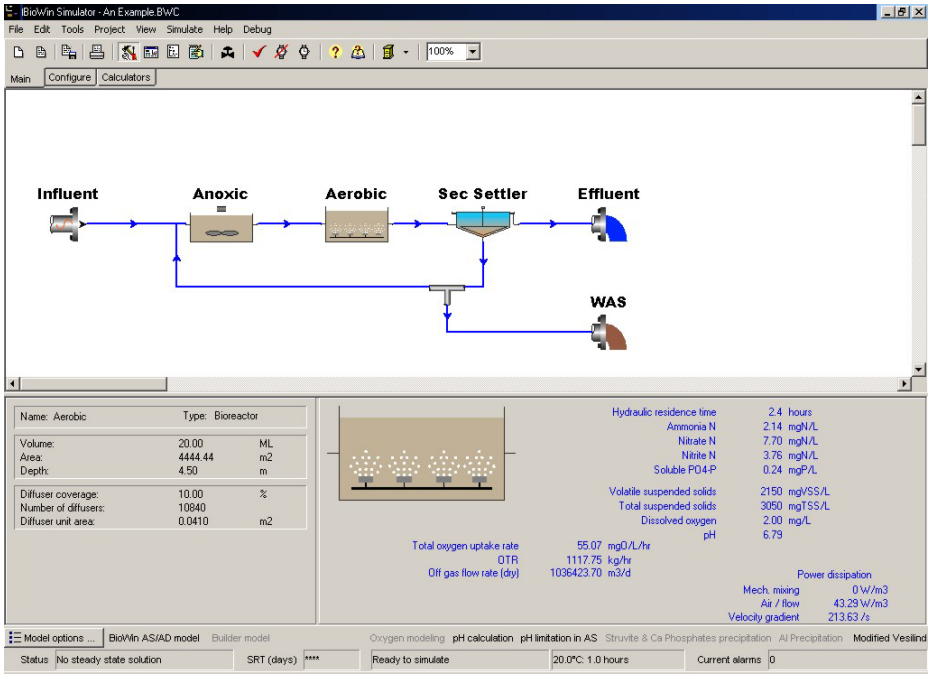
*Chart dialog box with clickable context sensitivity **?** in upper right hand corner*

Main Simulator Window

Parts of the Main Window

The BioWin main window consists of five main parts:

1. menus
2. toolbars
3. drawing board
4. summary panes
5. status bar



The main simulator window

The following sections give an outline of each part.

Main Window Menus

The various menus available in the main window are located at the top of the window. The menus available are:

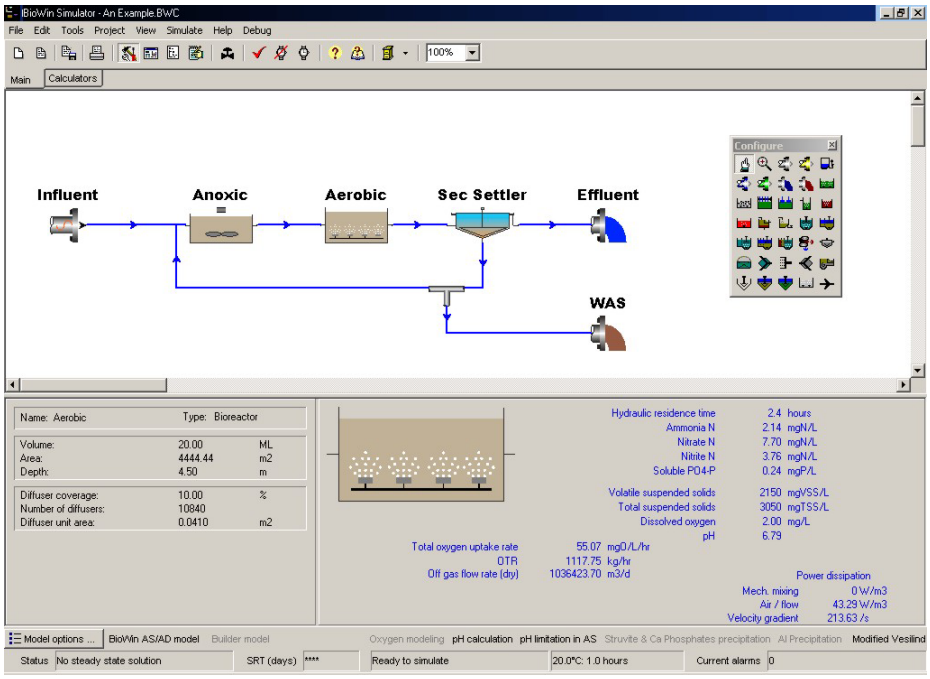
- File
- Edit
- Tools
- Project
- View
- Simulate
- Help

Each menu may be accessed with either of two methods:

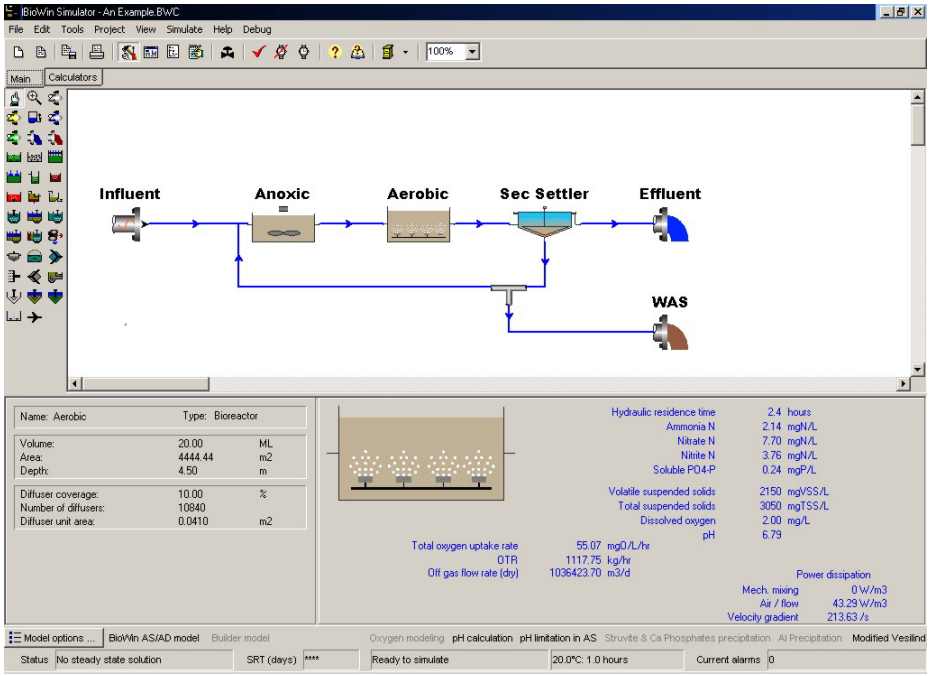
1. Click on the text of the menu, or;
2. Hold down the Alt key on your keyboard and press the letter on the keyboard corresponding to the underlined letter in the menu title. For example, Alt+F will access the File menu.

Toolbars

This section illustrates the various buttons found on the BioWin main window toolbars, and gives a description of each button's function. The BioWin toolbars can be toggled on/off with the menu command **Tools|Toolbars**. Note that the configure and calculator toolbars can also "float" over the drawing board in palette mode, or be docked in alternate locations (e.g. along the bottom or side of the drawing board). More information on the calculator toolbar may be found in **Solids Retention Time Calculation**in **Manage Projects**.



BioWin drawing board with floating toolbar in palette mode



BioWin drawing board with docked toolbar

Note : If a toolbar is closed while in palette mode, use the menu command **Tools|Toolbars** to get it back.

Main	Description
	New Configuration File: Click this button to close the current configuration and open a new one.
	Open Configuration File: Click this button to open a configuration file from disk.
	Save Configuration File: Click this button to save the current configuration file to disk.
	Print Configuration: Click this button to print a configuration using current printer settings.
	Show/Hide Configure Toolbar: Click this button to toggle the configure toolbar from visible to hidden.
	View BioWin Album: Click this button to open the BioWin album.
	View BioWin Explorer: Click this button to open the BioWin explorer.
	View Project Notes: Click this button to open the project notes editor.
	SRT Calculator/Control: Click this button to open to calculate or control the sludge age for a configuration.
	Check Simulation Data: Click this button to check that you have specified physical and operating data for all the elements in your configuration and that all elements in the configuration are properly connected with pipes.
	Steady State Solver: Click this button to open the steady state solver.
	Dynamic Simulation: Click this button to begin a dynamic simulation.

	Help on Help: Click this button to access help on the help system.
	Help Contents & Index: Click this button to access the help system.
	Pre-configured File Cabinet: Click the arrow next to this button to select and load pre-configured BioWin process files (e.g. various BNR configurations).
	Zoom Tool: Click this button to activate the zoom tool.
Configure	
	Element Selection Tool: Click this button to obtain the drawing board pointer to select/move/copy/edit process layout elements.
	Influent: Click this button to add an influent element to a configuration.
	Influent (BOD): Click this button to add a BOD influent element to a configuration.
	Methanol: Click this button to add a methanol addition stream to a configuration.
	Metal Addition: Click this button to add a metal stream (for chemical phosphorus removal) to a configuration.
	Influent (State Variable): Click this button to add a state variable influent element to a configuration.
	Effluent: Click this button to add an effluent element to a configuration.
	Sludge Output: Click this button to add a sludge output element to a configuration. The information displayed in the main window summary panes for this output are more relevant to a solids stream than a regular effluent (you may still use a regular effluent element for a solids stream if you wish).
	Bioreactor: Click this button to add a bioreactor element to a configuration.
	Brush Aerator Bioreactor: Click this button to add a brush aerator bioreactor element to a configuration.
	Surface Aerator Bioreactor: Click this button to add a surface aerator bioreactor element to a configuration.
	Variable Volume / Batch Bioreactor: Click this button to add a variable volume / batch bioreactor to a configuration.
	Aerobic Digester: Click this button to add an aerobic digester element to a configuration.
	Grit Tank: Click this button to add a grit tank element to a configuration.
	Equalization Tank: Click this button to add an equalization tank element to a configuration.
	Single Tank Sequencing Batch Reactor: Click this button to add a single tank sequencing batch reactor (STSBR) to a configuration.
	SBR + 1 Mix/Settle Prezone: Click this button to add an SBR + 1 mix/settle prezone element to a configuration.
	SBR + 1 Always-Mixed Prezone: Click this button to add an SBR + 1 always-mixed prezone element to a configuration.
	SBR + 2 Mix/Settle Prezones: Click this button to add an SBR + 2 mix/settle prezones element to a configuration.
	SBR + 2 Always-Mixed Prezones: Click this button to add an SBR + 2 always-mixed prezones element to a configuration.
	Dewatering Unit: Click this button to add a dewatering unit element to a configuration.

	Activated Primary Settler: Click this button to add an activated primary settling tank element to a configuration.
	Anaerobic Digester: Click this button to add an anaerobic digester element to a configuration.
	Sidestream Mixer: Click this button to add a sidestream mixer element to a configuration.
	General Mixer: Click this button to add a general mixer element to a configuration (the functionality of the two mixer styles is identical, however it is easier to connect multiple streams to a general mixer).
	Splitter: Click this button to add a splitter element to a configuration.
	Primary Settler: Click this button to add a primary settler element to a configuration.
	Point Settler: Click this button to add a point settler element to a configuration.
	Ideal Secondary Settler: Click this button to add an ideal secondary settler element to a configuration.
	Model Secondary Settler: Click this button to add a model secondary settler element to a configuration.
	Model Builder Unit: Click this button to add a model builder unit to a configuration.
	Pipe: Click this button to add a pipe element to a configuration.
100%	**Drawing Board Scale Setting:** Click the arrow to access a list of drawing board scale views, or enter a value directly.

Drawing Board

Replaces the Configure and Simulate windows of the previous BioWin versions.

The drawing board is the largest part of the BioWin main window. This is where you set up the process layout that you will be simulating by placing various elements, specifying their properties, and connecting them with pipes. A number of procedures can be carried out from the drawing board – some of the common ones are outlined in this section.

Place an Element on the Drawing Board

Element Placement Cursor

3. With the mouse cursor, click the button corresponding to the element you want on the configure toolbar.
4. Move the cursor onto the drawing board. When you do this, the cursor will change to the element placement cursor. Click on the drawing board where you want the element to be placed.
5. If you wish to place more of the same element type, continue clicking on the desired drawing board locations.
6. **Repeat steps 1-3** for all of the different element types in your configuration.

If after placing the element on the drawing board, you want to change the element's vertical or horizontal orientation:

Element Selection Tool

1. Click on the element selection tool from the configure toolbar
2. Right-click the element, and from the resulting popup menu, choose **Flip horizontal** or **Flip vertical** (the latter option only is available

for elements such as splitters and mixers).

Name an Element on the Drawing Board


Element Selection Tool


Element Selection Cursor

3. Click on the element selection tool from the **configure** toolbar.
4. Move the cursor over the element on the drawing board you wish to name. When you do so, the cursor will change to the element selection cursor.
5. Right-click on the element. In the resulting pop-up menu, select **Name**…
6. Enter the desired name for the element in the resulting dialog box and click **OK**.

Select Multiple Elements on the Drawing Board

To select multiple elements by dragging with the mouse:


Element Selection Tool

7. Click on the element selection tool from the **configure** toolbar.
8. Position the cursor above and to the left of the group of elements on the drawing board you wish to select.
9. Hold the mouse button and drag the cursor to a position below and to the right of the group of elements you wish to select.
10. Release the mouse button.

To select multiple elements by clicking with the mouse, while holding either the **Shift** or **Ctrl** key, click on each element of the group you wish to select.

Move an Element on the Drawing Board


Element Selection Tool


Element Selection Cursor

11. Click on the element selection tool from the configure toolbar.
12. Move the cursor over the element on the drawing board you wish to move. When you do so, the cursor will change to the element selection cursor.
13. Click on the element and while holding down the mouse button, drag the element to the desired new location.

Note: You also can move multiple elements simultaneously. Select the group of elements you wish to move, click on one of them, and drag the group to the desired new location.

Copy Elements on the Drawing Board


Element Selection Tool

14. Click on the element selection tool from the configure toolbar.


Element Selection Cursor

15. Move the cursor over the element you wish to copy. When you do so, the cursor will change to the element selection cursor.
16. Click and hold the mouse button.
17. While holding down the **Ctrl** key, drag the cursor to the location on the drawing board where you want the copy of the element to appear.
18. Release the mouse button.
19. Release the **Ctrl** key.

Note: You also can copy multiple elements simultaneously. Select the group of elements you wish to copy, position the cursor over one of the elements in the group and then proceed to step 3 above.

Delete an Element from the Drawing Board


Element Selection Tool


Selection Cursor

20. Click on the element selection tool from the configure toolbar.
21. Move the cursor over the element on the drawing board you wish to delete. When you do so, the cursor will change to the element selection cursor.
22. Click on the element and press the **Delete** key on your keyboard. Click **Yes** on the resulting confirmation dialog.
23. **Repeat steps 1-3** for the various elements in your configuration you wish to delete.

Note: You also can delete multiple elements simultaneously. Select the group of elements that you wish to delete, press the Delete key on your keyboard, and click Yes on the confirmation dialog.

Connect Elements with Pipes


Pipe Tool


Pipe Start


Pipe Crosshairs


No Pipe Allowed


Pipe End


No Pipe Allowed

24. Click the pipe tool on the configuration toolbar.
25. When you move the cursor onto the drawing board, the cursor will change to the "pipe start" cursor.
26. Place the cursor over the element area where you wish the pipe to start from.
27. If the location is appropriate, a set of crosshairs will appear on the "pipe start" cursor.
28. If the location is inappropriate, the cursor will change to a circle with a slash through it to indicate that a pipe may not begin at that location.
29. Click once and move the cursor to the desired location of the element where you wish the pipe to end, and click again.
30. As you move the pipe towards the element where you wish it to end, the cursor will change to the "pipe end" cursor.
31. If the location of the pipe terminus is appropriate, this cursor will remain.

32. If the location is inappropriate, the cursor will change to a circle with a slash through it to indicate that a pipe may not end at that location.
33. Repeat steps 3-9 until you have connected all your elements with pipes.

Access Element Properties from the Drawing Board


Element Selection Tool


Selection Cursor

34. Click on the element selection tool from the configure toolbar.
35. Move the cursor over the element on the drawing board you wish to view the properties of. When you do so the cursor will change to the element selection cursor.
36. Right-click on the element and select **Properties**… in the resulting pop-up menu, or double-click on the element.

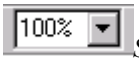
Note: You may also access an element's properties by double-clicking the element.

Zoom In on a Drawing Board Area


Zoom Tool


Zoom Cursor

37. Click on the zoom tool from the configure toolbar.
38. Move the zoom cursor to a position above and to the left of the area on the drawing board that you wish to zoom in on.
39. Click and drag the cursor down and to the right of the area you wish to zoom in on.
40. Release the mouse button to finish.

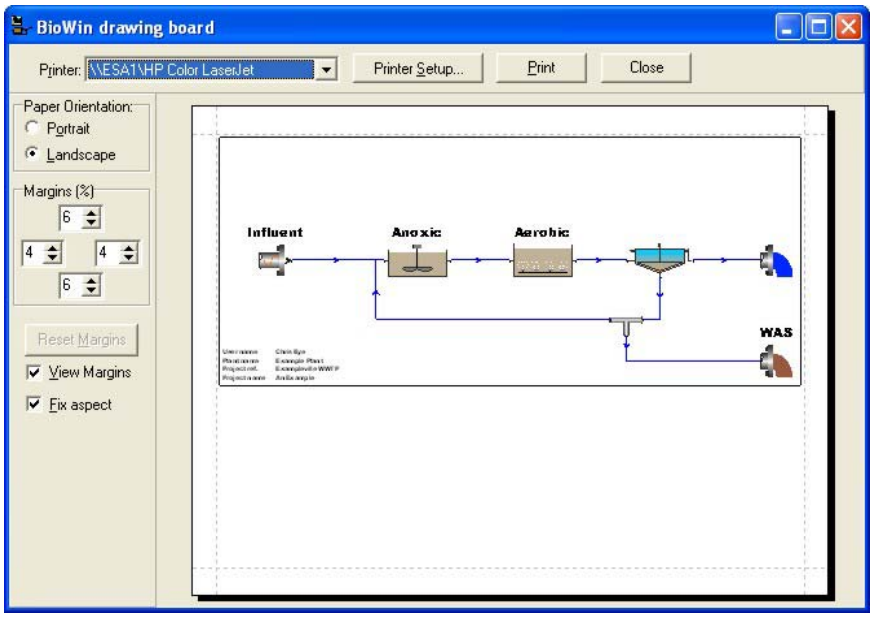


Scale Setting

You may also zoom in by selecting or typing in a desired zoom percentage setting on the configure toolbar.

Printing the Drawing Board

You can print out the BioWin drawing board using the **File|Print Flowsheet…** command. This command will invoke the print drawing board dialog box, which is shown below:



Dialog box used for printing the drawing board

Use the **Printer** drop list box to select the printer you want to use for printing. The **Printer Setup…** button will open the printer setup dialog box which will allow you to access the printer's properties, set paper size, page orientation, and a number of other printer options (the options presented to you will be dependant on the printer you have). The **Print** button will send the print job to the printer and the printout will match the preview that is shown. The **Close** button closes this dialog box and returns you to the drawing board.

Using the **Paper Orientation** group, specify whether you want the printing to be done on a **Portrait** or **Landscape** page. The print preview gives you an idea of what the printout will look like under each format.

If you do not wish to see the size of the margins for your print job, you may de-select the box labeled **View Margins**. You can control the margins using three different methods:

41. Using the **Margins (%)** spin edits, you can adjust each margin as you like. The four spin edit boxes each control the margin that shares its position, that is, the top spin edit controls the top margin, the bottom spin edit controls the bottom margin, and so on. When you change a value, you will see changes in the print preview accordingly.

Resize Cursors

42. You may drag each margin using the mouse. Position the mouse cursor over the margin you wish to adjust until the horizontal or vertical resize cursor appears. Click the mouse button, hold it, and drag the margin to the position you wish it to occupy. Notice that when you finish dragging it, the values in the **Margins (%)** spin edits will have been updated.

Drag Print Object

43. By moving the object to be printed around on the page. When the mouse cursor takes the form of a hand, you may click and drag the entire object around on the page until it is in the desired position. Notice that when you finish dragging it, the values in the **Margins (%)** spin edits will have been updated.

If after applying any one of these methods of adjusting margins you wish to reset the margins to the default values, you may do so by clicking the **Reset Margins** button.

If you want the printed picture of the drawing board to have the same length and width proportions as the actual drawing board, then select the box labeled **Fix aspect**.

Main Window Summary Panes

The purpose behind the main window summary panes is to provide a brief amount of information about various elements that have been placed on the drawing board. Two summary panes are found at the bottom of the main simulator window. The left pane contains a small table that lists an element's name, type, and its physical dimensions. The right pane contains a picture of the element and detailed, element-specific information about various compounds and parameters.

View Element Information in the Summary Panes


Element Selection Tool

44. Click on the element selection tool from the **configure** toolbar.
45. Hold the cursor over the element that you wish to view summary information about.

Note: Information also will be displayed in the summary panes if you hold any one of the "pipe" cursors over an element. Also note that the airflow per diffuser figure shown in the right summary pane for SBR elements is for the main zone only.

It also is possible to "freeze" the summary panes so that they display information about a specific element regardless of where you move your cursor. To do this, follow these steps:

46. Hold your mouse cursor over either the left or right pane, and right-click.
47. From the small popup menu that appears, choose Select…
48. A dialog box will open – from the Elements drop list box, select the element in your configuration that you want to "freeze" the summary panes on.
49. Click the Close button to finish. You should now be able to move the mouse cursor around your configuration without the summary panes changing.

If at any time you want to return the summary panes to the mode where they display information on the element your mouse cursor is held over, simply right-click on either of the panes and choose Fly by from the small popup menu that appears.

Resize the Summary Panes

To change the horizontal dimensions of the summary panes:


Horizontal Resize

50. Position the cursor over the vertical line that divides the two panes such that the horizontal resize cursor appears.
51. When the horizontal resize cursor appears, click and drag the cursor until the panes have the desired widths.

To change the vertical dimensions of the summary panes:

Vertical Resize

52. Position the cursor at the top of the summary panes, just below the drawing board horizontal scroll bar such that the vertical resize cursor appears.
53. When the vertical resize cursor appears, click and drag the cursor until the panes have the desired heights.

Main Window Status Bar

The status bar at the bottom of the BioWin main window displays two main classes of information – simulation status information and model details information.

Simulation Status Information

The status bar at the bottom of the main window operates in two modes; different information is displayed in each mode.

In the first mode, the status bar provides a brief hint or description of toolbar buttons and menu items if the mouse cursor is held over them.

In the second mode (i.e. when the cursor is not held over a toolbar button or menu item), the status bar displays information about the status of the configuration, and whether a steady state solution has been found for the configuration:

- When you are adding elements to a configuration, the status bar will display the message **Building configuration**.
- When you have connected all the elements with pipes, the status bar will display the message **Configuration complete**.
- Once you have specified physical and operating data for all of the elements in the configuration, the status bar will display the message **Ready to simulate**.
- During each of these phases, the status bar also will display the message **No steady state solution**.
- Once you have found a steady state solution, the status bar will display the message **Steady state solution**.
- If you have set up SRT calculation for your project (see **Solids Retention Time Calculation** in "*Managing BioWin Projects*"), the **SRT (days)** section of the status bar will display the steady state SRT in days.
- If you are performing a dynamic simulation and you have set up SRT calculation, the **SRT (days)** section of the status bar will display an "instantaneous" SRT (denoted by an asterisk) in days. This is calculated based on the current mass and wastage rate in your system. To increase the update frequency of the status bar, see **Data Interval** in "*Managing BioWin Projects*".
- The fourth portion (moving from left to right) of the status bar displays the global temperature and the current database interval separated by a colon. For example: the following display indicates that the global temperature is 25 degrees Celsius, and the database interval is 2 hours.

  25.0 °C: 2.00 Hours

- The fifth portion (moving from left to right) of the status bar will display the number of alarms that have been triggered for the current

configuration. For more information on alarms, please see the alarm topics in the **Customizing BioWin** and **Managing BioWin Projects** sections.

Model Details Information

This section of the status bar contains information about the models being used in the current project:

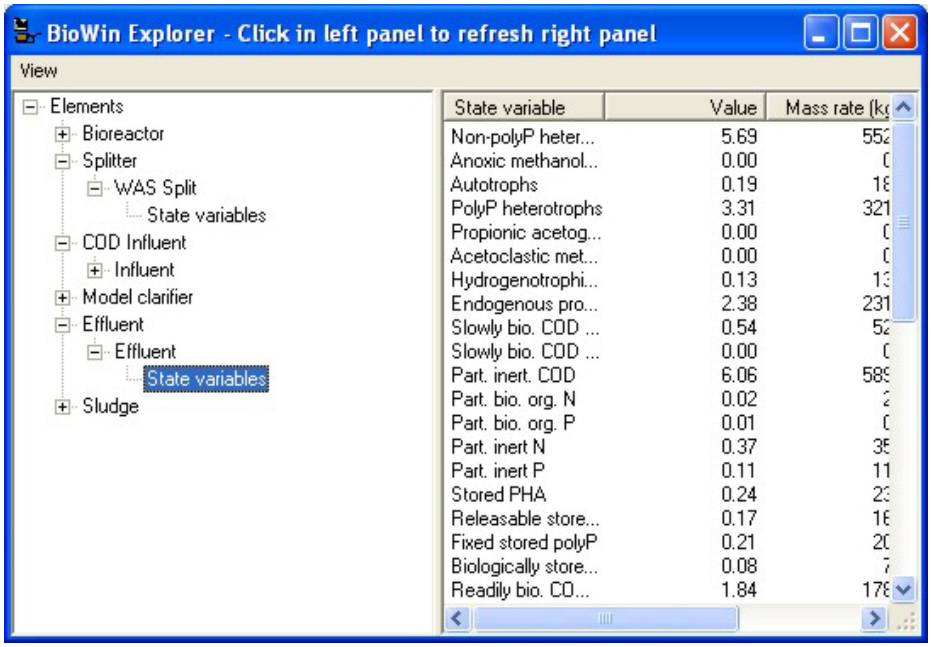
- At the left end of the status bar is a button labeled **Model options…**Clicking this button offers rapid access to the model options being used in the current project (see **Model Options**).

As you move from left to right a number of model details are given (note that if an option is active, the status bar text is bold, not grayed out):

- The next section notifies you whether the BioWin integrated activated sludge / anaerobic digestion model or a user-defined builder model is being used.
- Moving further to the right, the next section notifies you if oxygen modeling project is toggled on or off. (See reference to Oxygen Modeling in **Model Options**).
- The next section tells you if the activated sludge kinetic equations are incorporating pH inhibition effects.
- The next section notifies you if spontaneous precipitation of Struvite, HDP, and HAP are being modeled.
- The next section shows whether chemical phosphorous removal is being modeled.
- The next section tells you which one-dimensional settler model is being used in model settlers

BioWin Explorer

The BioWin explorer provides a means to display information about the elements that make up a configuration. You access the explorer from the main simulator window via the menu choice **View|Explorer**. From the explorer, you may return to the main simulator window or open the BioWin album using the respective menu commands **View|Main** and **View|Album**. The explorer window is split into two main panes, as shown in the picture below. The left pane displays an expandable/collapsible outline of the elements that make up the current configuration. The right pane displays information related to the selected node of the outline.



The BioWin explorer

Note : The information contained in the right pane will be updated each time you click on any branch of the expandable/collapsible outline in the left pane. Since the information displayed in the BioWin explorer is not updated automatically, you should always click on the branch that you are viewing to ensure that the information is current before recording or reporting it.

Navigating in the Explorer

There are different ways to move about in the explorer; the method you choose is a matter of preference only as the various techniques accomplish the same goals. A small box beside a node indicates that node contains sub-nodes. If there is a "+" sign in the box, this indicates that the node may be expanded to display its sub-nodes. To expand the node, you may either click on the "+" sign, or double-click the node title (i.e. the text immediately to the right of the box). If there is a "-" sign in the box, this indicates that the node currently is expanded and displaying all its sub-nodes. To collapse the node, you may either click on the "-" sign or double-click the node title.

When the explorer initially is opened, the top node entitled "Elements" is displayed in the left pane. In the first column of the right pane, all the elements that make up the current configuration are listed, grouped by element type and sorted within each group by the order in which they were added to the configuration. The remaining columns in the right pane contain the current values of a number of compounds and variables (the information displayed here can be changed – see **Explorer Options**). Double-clicking on a row in this list will open up the properties dialog box for the element in question and allow you to edit physical, operational, and monitoring settings. This is equivalent to right-clicking the element on the drawing board and choosing **Properties…** from the resulting pop-up menu, or double-clicking the element icon on the drawing board.

Expanding the "Elements" node shows all of its sub-nodes; these consist of a node for each of the different element types in the configuration. Clicking on an element-type node causes the right pane to display a list of the elements of that type that

currently are in the configuration in the first column, with current values of a number of compounds and variables in subsequent columns. Double-clicking on a row in this list will have the same effect as described above.

Clicking on an individual element node causes the right pane to display current values of a number of compounds and variables for that element. Double-clicking this row will allow you to access the element's properties. Expanding an individual element node will display the **State variables** node and, if appropriate, the **Parameters** node.

Clicking on the **State variables** node for an element causes the right pane to display a list of the current values of all state variables for that element, as well as the mass rate for each. Clicking on the **Parameters** node for an element causes the right pane to display a row of the local model parameters for that element. Double-clicking this row will open the **Model parameter editor** to allow these local model parameter values to be edited.

Explorer Appearance

You can adjust the width of the explorer columns in the right pane in two ways:

Horizontal Resize

- Hold your cursor over the right dividing line of the column you want to resize in the column heading row. When you do this, your cursor will change to the horizontal resize cursor. Click the mouse and drag the column dividing line until the column is the desired size
- To size a column so that it fits the widest value displayed in that column, simply click on the column heading. Note that this will undo any resizing you have done with method (1) as it sets the other columns to a "standard" width.

It also is possible to change the relative width of the two main explorer panes. When you hold your cursor over the vertical bar that separates the left and right panes, you will see the horizontal resize cursor appear. If you click and hold the mouse button, you will be able to drag the bar to the left or right, which will change the relative widths of the left and right panes.

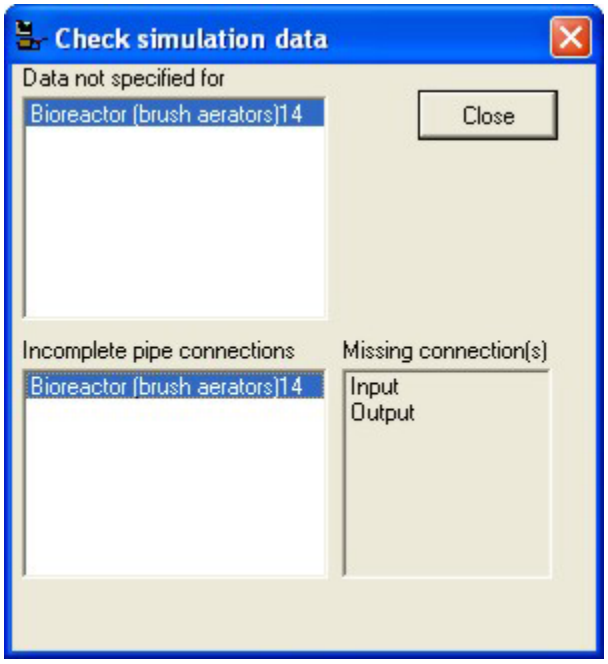
Running Simulations in BioWin

This section provides detailed information about the menu commands available in the **Simulate** menu of the main simulator window.

Note : For systems that are difficult to solve, see **Tips for Systems that are Difficult to Solve** in "*Managing BioWin Projects*".

Check Simulate Data

Use this command to ensure that you have properly specified data for all the elements in your configuration, and that all the elements in the configuration are connected with pipes. If you have elements in your configuration that are not connected with pipes, the other commands in the **Simulate** menu will be grayed out, and when you choose **Check simulate data**, the following dialog box will appear to notify you which elements are not properly connected. The elements that are not connected will be listed in the **Incomplete pipe connections** list, and if you click on an element in this group, the missing connection will be displayed in the **Missing connection(s)** list.



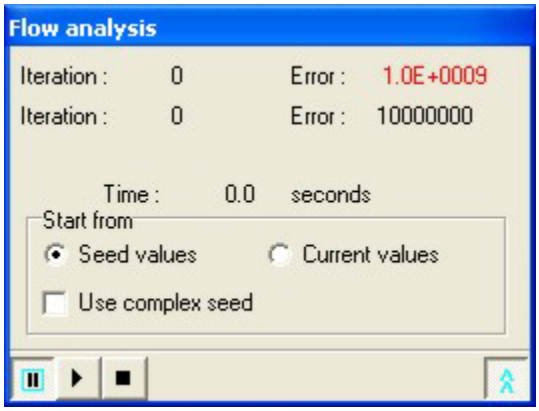
Dialog box used for checking simulation data

Note : Invoking this command also shows you the **Data not specified for** list which contains the names of elements that you have not yet specified physical and operating parameters for. Even if you wish to use the default values, you must acknowledge this by viewing the properties of the element if you wish to remove them from the list. You could do this by closing the dialog box, and accessing the properties for the elements in the list from the drawing board. However, a quicker method is to double-click on element names in the **Data not specified for** list. This will open the properties dialog box for the element you double-clicked on. Once the properties dialog is closed, you will be returned to the **Check simulate data** dialog box and the element that you double-clicked on will have been removed from the list.

Once you have specified data and pipe connections for all elements, all lists in the **Check simulate data** dialog box will be empty. You may then invoke other commands in the **Simulate** menu.

Flow Balance

This command will open the flow solver dialog box, seen below. Note that if you have not specified physical and operating data for the elements in your configuration, you will be presented with the **Check simulation data** dialog box – you can bypass this warning by clicking the **Accept** button.



Simulator controls and information display

Clicking the **Pause** button will suspend the flow balance solver at the current time shown.

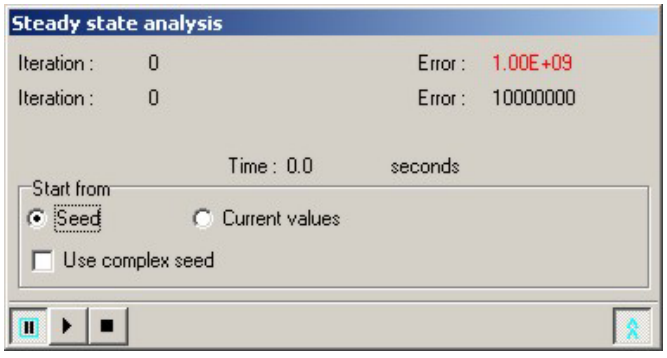
Clicking the **Play** button either begins the flow balance solver or resumes it if the **Pause** button has been clicked.

Clicking the **Stop** button will terminate the flow balance solver at the current time shown.

Clicking the **Details** button shows / hides the flow balance solver information. The information displayed consists of the iteration number of the two most recent solver iterations, the error of each iteration (i.e. the maximum difference between the current iteration values and the steady state solution values), and the time that has elapsed between the two most recent iterations.

Steady State Balance

This command will open the steady state solver dialog box, seen below. Note that if you have not specified physical and operating data for the elements in your configuration, you will be presented with the **Check simulation data** dialog box – you can bypass this warning by clicking the **Accept** button.



Simulator controls and information display

Clicking the **Pause** button will suspend the steady state solver at the current time shown.

Clicking the **Play** button either begins the steady state solver or resumes it if the **Pause** button has been clicked.

Clicking the **Stop** button will terminate the steady state solver at the current time shown.

Clicking the **Details** button shows / hides the steady state solver information. The information displayed consists of the iteration number of the two most recent solver iterations, the error of each iteration (i.e. the maximum difference between the current iteration values and the steady state solution values), and the time that has elapsed between the two most recent iterations.

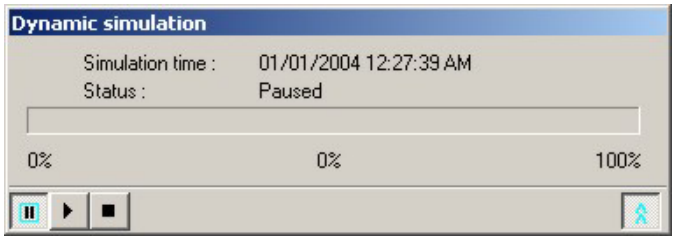
The **Start from** group contains three options for controlling the starting conditions that the steady state solver uses in obtaining a solution. If you select **Seed values**, BioWin will use its seeding algorithm to place numerical seed values in all the elements of a configuration. The seeding algorithm is based on various factors including the influent loading and the seed sludge age specified on the **Numerical parameters** tab accessed via the **Project|Current Project Options** menu command. If you select **Current values**, whatever numerical values exist in a configuration's elements will be used as steady state solver starting values, i.e. BioWin will not use its seeding algorithm. This option can be useful if you have already solved a steady state on a complex system and a minor change requiring a new solution has been made. Quite often starting from a steady state rather than re-seeding will result in a quick solution. If the **Use complex seed** is selected, the following sequence of events takes place:

- The configuration is seeded using the average loading conditions and the user-specified seed sludge age.
- BioWin begins to run a dynamic simulation using the average loading conditions. Two progress bars display information about the dynamic simulation – the top bar shows the progress towards the maximum dynamic simulation time of 20 days, and the bottom bar indicates the "stability" of the dynamic simulation.
- When one of these criteria is met, the BioWin steady state solver will begin its solution routine using the current values for seeding. Note that the user may interrupt this process and force the steady state solver to begin with whatever current values are present before one of the two criteria has been met.

Note : The right portion of the main simulator window status bar contains information about whether or not a steady state solution has been found for the current configuration.

Dynamic Simulation

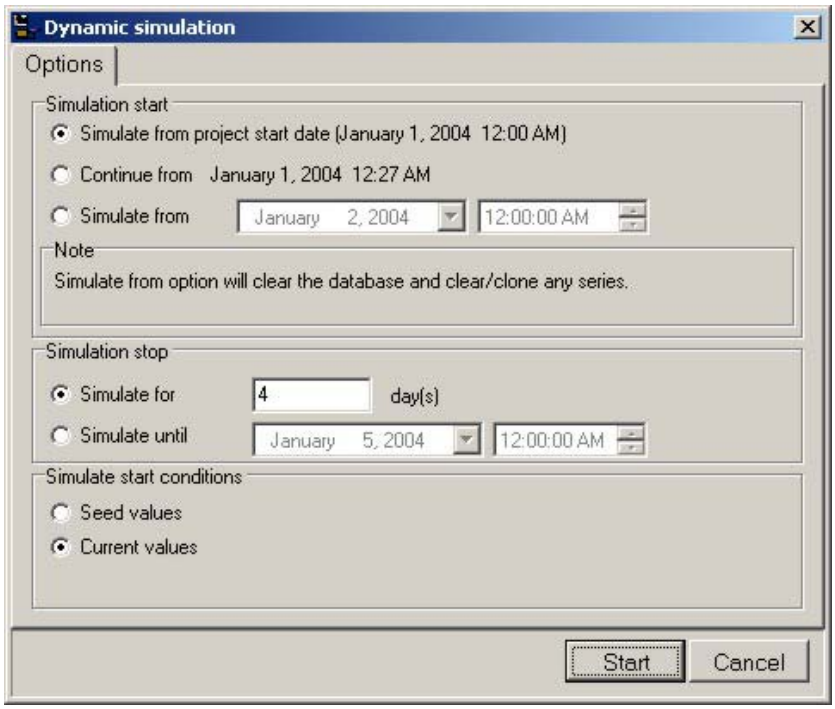
This command will open the dynamic simulation dialog box, seen below. Note that if you have not specified physical and operating data for the elements in your configuration, you will be presented with the **Check simulation data** dialog box – you can bypass this warning by clicking the **Accept** button.



Simulator controls and information display

Clicking the **Pause** button will suspend the dynamic simulation at the current time shown.

Clicking the **Play** button either begins a dynamic simulation or continues one if the simulator has been paused (either by pressing the **Pause** button or by reaching the end of a previous dynamic simulation). If the **Play** button is clicked to begin a dynamic simulation, the following dialog box will be seen:



Dialog used to begin a dynamic simulation

In the **Simulate start** group, you choose the time at which the dynamic simulation will begin:

- If you select Simulate from project start date then 12:00 AM on the day that you specified as the simulation start date on the Project information tab accessed from the Project|Info menu command will be used.
- If you select Continue from then the time at which the previous dynamic simulation was stopped will be used.
- If you select Simulate from, then you may specify the exact day (using the calendar that appears when the drop arrow is clicked) and time (using the spin edit box) at which the dynamic simulation will start. Note that if this option is chosen, then the current contents of the database will be cleared and any time series charts will be cleared or saved in the album (depending on the options you have set on the **General** tab in the **Tools|Customize** menu).

In the **Simulation stop** group, there are two methods for specifying the length of the dynamic simulation:

- If you select **Simulate for**, then you may specify the length of the dynamic simulation in days.
- If you select **Simulate until**, then you may specify the exact length of the dynamic simulation by choosing the day (using the calendar that

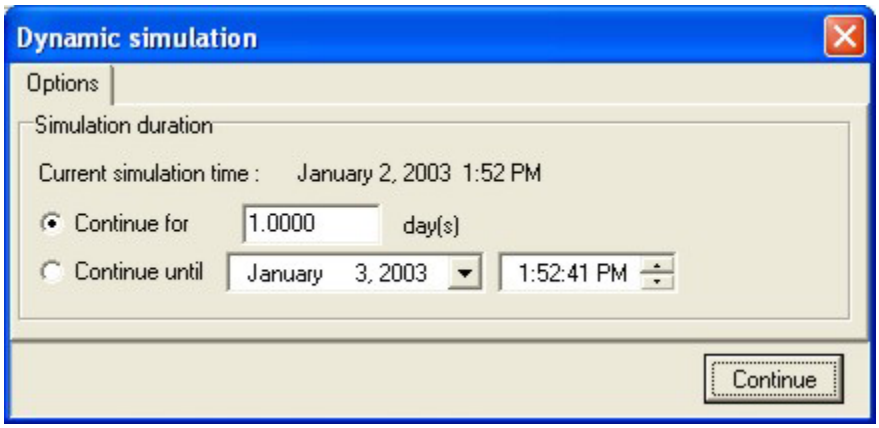
appears when the drop arrow is clicked) and time on that day (using the spin edit box) at which the dynamic simulation will end.

The **Simulate start conditions** group contains options that pertain to numerical seeding at the start of the dynamic simulation:

- If you select **Seed values**, the simulator will be seeded with default state variable values.
- If you select **Current values**, then the simulator will be seeded with the current values of state variables.

Clicking **Start** will begin the dynamic simulation using your specified choices.

If the **Play** button is clicked to restart a dynamic simulation after the simulator has been paused, the following dialog box will be seen:



Dialog used to continue a dynamic simulation

In the **Simulation duration** group, you can specify the ending time of the dynamic simulation:

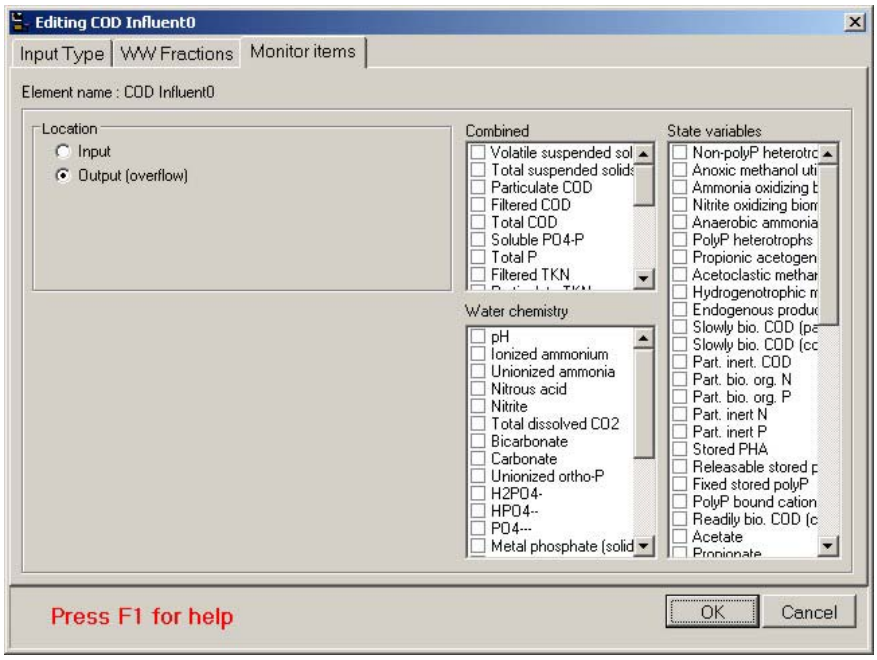
- If you select **Continue for**, then you may specify the length of the dynamic simulation in days. If you want the simulation to end at the time you specified at the beginning of the simulation (i.e. you simply want to continue after pausing mid-simulation), do not change this value.
- If you select **Continue until**, then you may specify the exact length of the dynamic simulation by choosing the day (using the calendar that appears when the drop arrow is clicked) and time on that day (using the spin edit box) at which the dynamic simulation will end. If you want the simulation to end at the time you specified at the beginning of the simulation (i.e. you simply want to continue), do not change this value.

Clicking **Continue** will resume your dynamic simulation.

Element Types

Influent

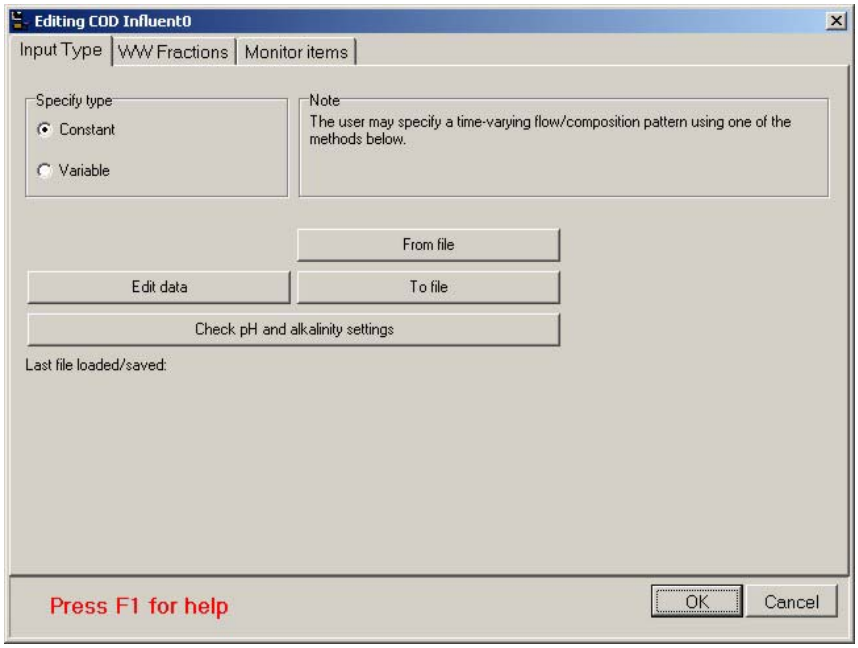
The influent element is used to control the wastewater flow/composition to your process configuration. When you add an influent element to your configuration, you can control the type (i.e. constant or variable) of influent and the wastewater fractions. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The influent monitor items tab

Influent Type

The **Influent type** tab, shown in the figure below, is used to specify the influent element as one of two possible types: **constant** or **variable**. A constant influent's wastewater flow/composition is constant during a steady state simulation, but may be modified by the user during a dynamic simulation. The wastewater flow/composition of a constant influent may be changed during a dynamic simulation only if the simulator is paused.



The influent type tab

A variable influent element represents a user-specified time-varying flow / composition stream entering the system. For steady state calculations the mass average of the time-varying flow / composition pattern is used. Each set of conditions has an associated starting time; this time is the dynamic simulation time at which these conditions will start to be used. The scheduled influent places the following restrictions on the starting time for each set of conditions:

1. The starting time for each set of conditions must be greater than the starting time for the previous set.
2. The first set of conditions must start at time 0.
3. The starting time of the final set of conditions must be less than the time specified in the **Cycle duration** edit box.

During a dynamic simulation each set of influent conditions is used from when the simulation time equals its starting time until the simulation time reaches the starting time of the following set of conditions. The final set of influent conditions is used from its starting time until the **Cycle duration** time is reached. At the end of the cycle the process is repeated using the **Cycle duration** time as time zero.

Clicking the **Edit data** button will present you with a dialog that is dependant on the type of influent. If the influent is constant, you will be presented with the **Edit influent** dialog box. If the influent is scheduled, you will be presented with the **Edit influent itinerary** dialog box. For more information on the use of these dialog boxes, please see **Itinerary Editors**

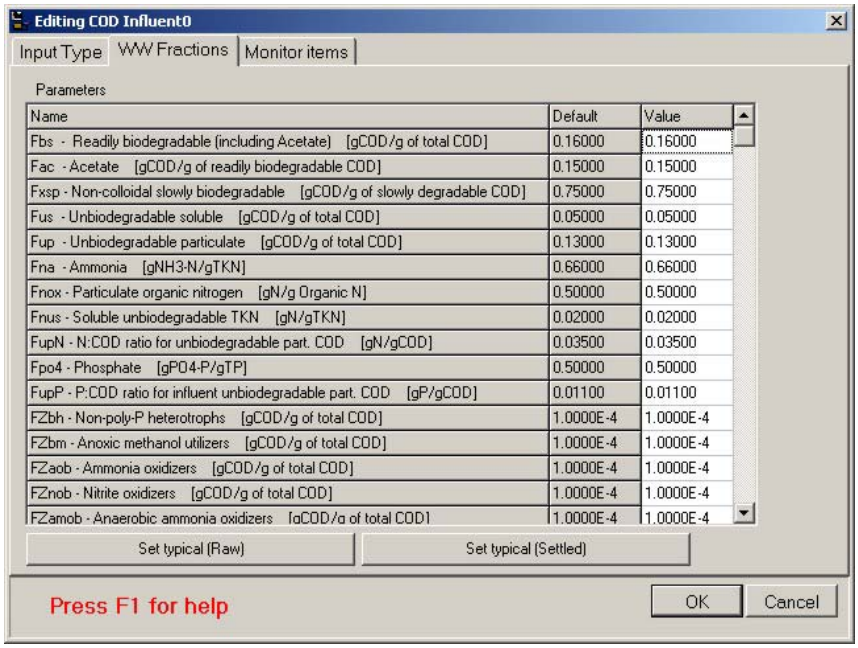
Clicking the **Check pH and alkalinity settings** button will check that the pH and alkalinity values that you have entered are consistent. That is, that BioWin is able to provide an acceptable mix of dissolved carbon dioxide, anions and cations to match the settings you have specified. This process is recommended although it can be time consuming for large influent itineraries (e.g. greater than one thousand rows). Should you decide not to do it then BioWin will generate an alarm if the condition is encountered during a simulation.

You also may input data by loading a previously saved data file. Clicking the **From file** button will present you with the **Open influent file** dialog box. Using this dialog box, select the influent data file you want and click **Open** to finish.

You also may save a data file if you feel that you may want to use it again. Clicking the **To file** button will present you with the **Save influent file** dialog box. Using this dialog box, save the data file under a name and location that you will remember.

Wastewater Fractions

The **Wastewater fractions** tab, shown in the figure below, is used to specify the fractional composition of the influent wastewater.



The influent wastewater fractions tab

You may specify the following fractional values:

Name	Description
F_{bs}	Fraction of total influent COD which is readily biodegradable [(S_{bsc} + S_{bsa}) / Total influent COD]
F_{ac}	Fraction of readily biodegradable COD which is VFAs [S_{bsa} / (S_{bsa}+S_{bsc})]
F_{xsp}	Fraction of slowly biodegradable influent COD which is particulate [X_{sp} / (X_{sc} + X_{sp})]
F_{us}	Fraction of total influent COD which is soluble unbiodegradable
F_{up}	Fraction of total influent COD which is particulate unbiodegradable
F_{na}	Fraction of influent TKN which is ammonia
F_{nox}	Fraction of influent biodegradable organic nitrogen which is particulate
F_{nus}	Fraction of influent TKN which is soluble unbiodegradable

F_{po4}	Fraction of influent TP which is phosphate
F_{upN}	The N:COD ratio for the influent particulate unbiodegradable COD
F_{upP}	The P:COD ratio for the influent particulate unbiodegradable COD
FZ_{bh}	Fraction of total influent COD which is non-polyP heterotrophic organisms.
FZ_{ba}	Fraction of total influent COD which is autotrophic organisms.
FZ_{aob}	Fraction of total influent COD which is ammonia oxidizing organisms.
FZ_{nob}	Fraction of total influent COD which is nitrite oxidizing organisms.
FZ_{amob}	Fraction of total influent COD which is anaerobic ammonia oxidizing organisms.
FZ_{bp}	Fraction of total influent COD which is polyP heterotrophic organisms.
FZ_{bpa}	Fraction of total influent COD which is propionic acid acetogen organisms.
FZ_{bam}	Fraction of total influent COD which is acetoclastic methanogen organisms.
FZ_{bhm}	Fraction of total influent COD which is H_2-utilizing methanogen organisms
FZ_{bm}	Fraction of total influent COD which is anoxic methanol utilizing organisms

Note: If you enter values for FZ_{bh}, FZ_{aob}, FZ_{nob}, FZ_{amob}, FZ_{ba}, FZ_{bp}, FZ_{bpa}, FZ_{bam}, FZ_{bhm}, or FZ_{bm} remember that organisms contain nitrogen and phosphorus.

An explanation of the fractionation of influent nitrogen may be helpful at this point. Ammonia is given by:

$$NH_3 = F_{na} \cdot TKN_t$$

Soluble unbiodegradable organic nitrogen is given by:

$$N_{US} = F_{nus} \cdot TKN_t$$

Nitrogen from organisms present in the influent is calculated by the sum of the products of the various organism concentrations and their respective nitrogen fractions, i.e.:

$$OrganismsN = \sum Zb\chi \cdot f_{N,ZB\chi}$$

Unbiodegradable particulate nitrogen is given by:

$$X_{IN} = F_{up,N} \cdot F_{up} \cdot COD_t$$

The remaining organic nitrogen is broken into particulate and soluble components. Particulate biodegradable organic nitrogen is given by:

$$X_{ON} = (TKN_t - NH_3 - N_{US} - X_{IN} - OrganismsN) \cdot F_{nox}$$

Soluble biodegradable organic nitrogen is given by:

$$N_{OS} = (TKN_t - NH_3 - N_{US} - X_{IN} - OrganismsN) \cdot (1 - F_{nox})$$

Similarly, an explanation of the fractionation of influent phosphorus is as follows. Soluble orthophosphate is given by:

$$PO_4 = F_{po4} \cdot TP$$

Phosphorus from organisms present in the influent is calculated by the sum of the products of the various organism concentrations and their respective phosphorus fractions, i.e.:

$$OrganismsP = \sum Zb\chi \cdot f_{P,ZB\chi}$$

Unbiodegradable particulate phosphorus is given by:

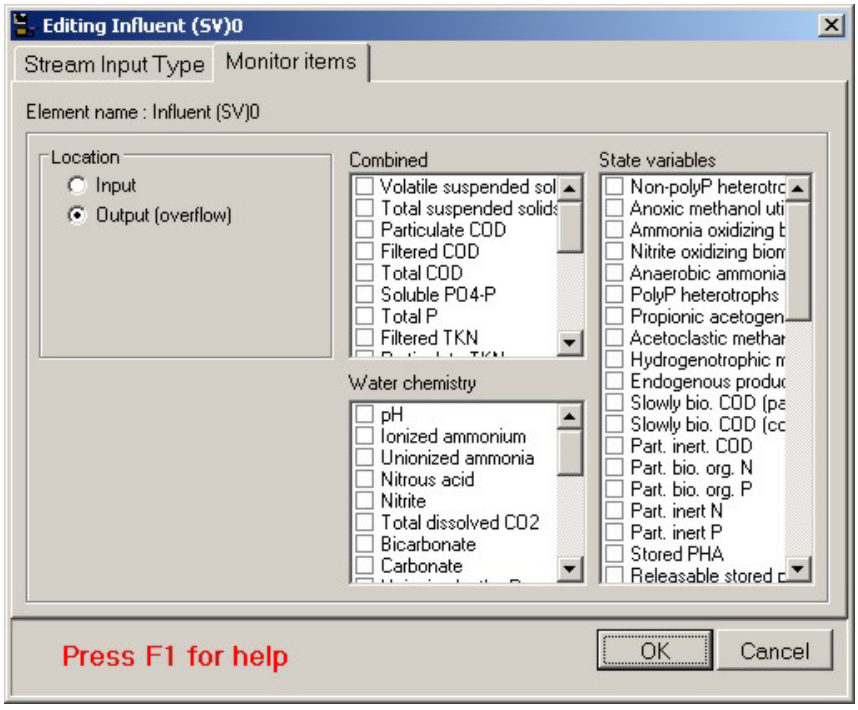
$$X_{IP} = F_{up,P} \cdot F_{up} \cdot COD_t$$

The remaining particulate biodegradable organic phosphorus is given by:

$$X_{OP} = TP - PO_4 - X_{IP} - OrganismsP$$

Influent (State Variable)

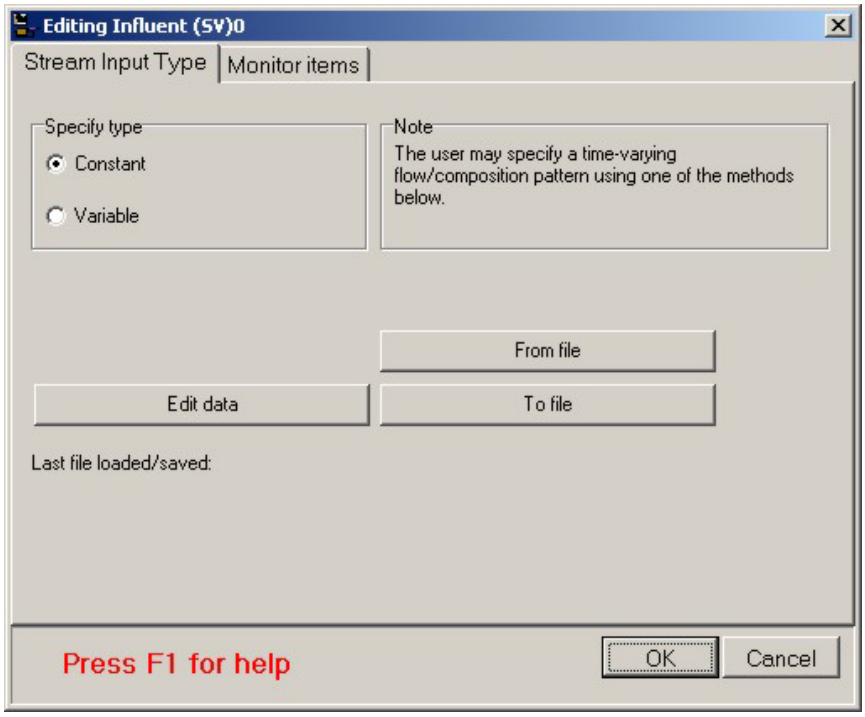
The influent (state variable) element is used to control the wastewater flow/composition to your process configuration. **The influent (state variable) element differs from the standard influent element in that its composition is specified in terms of state variable concentrations, rather than total concentrations (e.g. COD, TKN) and fractions (e.g. F_{UP}, f_{NA}).** For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The influent (state variable) monitor items tab

Influent (State Variable) Type

The **Influent (state variable) type** tab, shown in the figure below, is used to specify the state variable influent element as one of two possible types: **constant** or **variable**. A constant state variable influent's wastewater flow/composition is constant during a steady state simulation, but may be modified by the user during a dynamic simulation. The wastewater flow/composition of a constant state variable influent may be changed during a dynamic simulation only if the simulator is paused.



The influent (state variable) type tab

A variable state variable influent element represents a user-specified time-varying flow / composition stream entering the system. For steady state calculations the mass average of the time-varying flow / composition pattern is used. Each set of conditions has an associated starting time; this time is the dynamic simulation time at which these conditions will start to be used. The variable state variable influent places the following restrictions on the starting time for each set of conditions:

1. The starting time for each set of conditions must be greater than the starting time for the previous set.
2. The first set of conditions must start at time 0.
3. The starting time of the final set of conditions must be less than the time specified in the **Cycle duration** edit box.

During a dynamic simulation each set of state variable influent conditions is used from when the simulation time equals its starting time until the simulation time reaches the starting time of the following set of conditions. The final set of state variable influent conditions is used from its starting time until the **Cycle duration** time is reached. At the end of the cycle the process is repeated using the **Cycle duration** time as time zero.

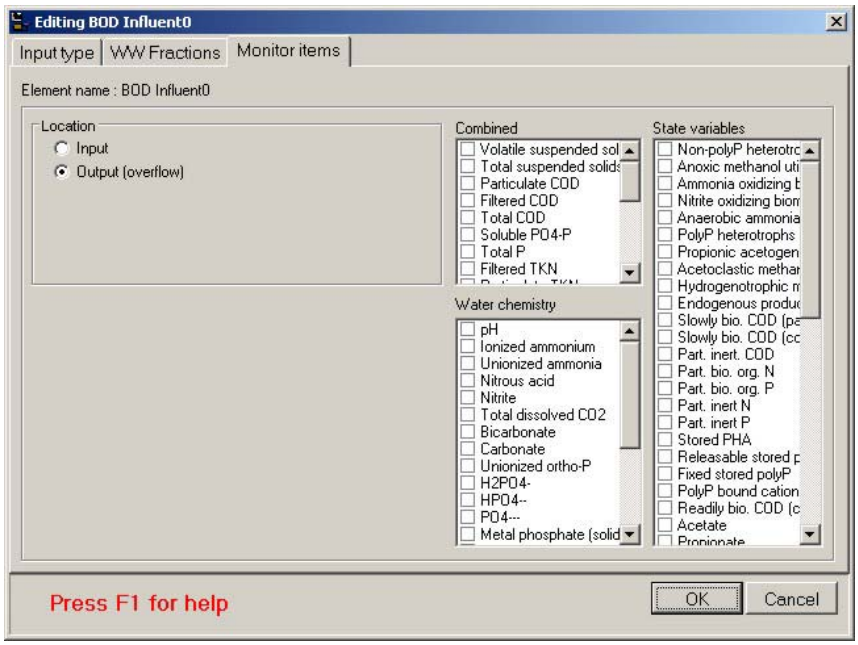
Clicking the **Edit data** button will present you with a dialog that is dependant on the type of state variable influent. If the state variable influent is constant, you will be presented with the **Edit influent** dialog box. If the influent is scheduled, you will be presented with the **Edit influent itinerary** dialog box. For more information on the use of these dialog boxes, please see **Itinerary Editors**.

You also may input data by loading a previously saved data file. Clicking the **From file** button will present you with the **Open influent file** dialog box. Using this dialog box, select the influent data file you want and click **Open** to finish.

You also may save a data file if you feel that you may want to use it again. Clicking the **To file** button will present you with the **Save influent file** dialog box. Using this dialog box, save the data file under a name and location that you will remember.

Influent (BOD)

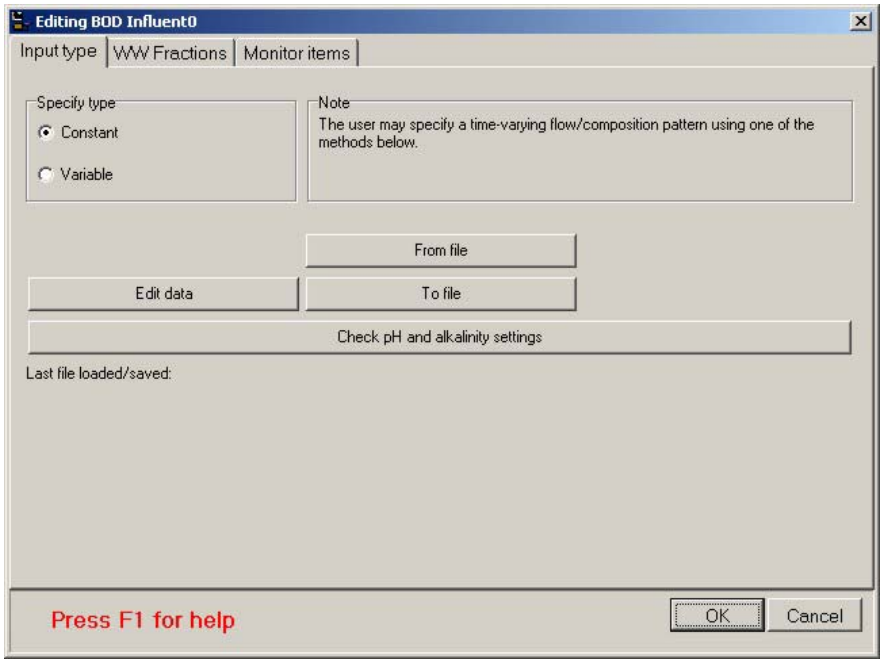
The BOD influent element is used to control the wastewater flow/composition to your process configuration. **The BOD influent element differs from the standard influent element in that its organic strength is specified in terms of BOD concentration, rather than COD concentration.** For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The BOD influent monitor items tab

Influent (BOD) Type

The **BOD Influent type** tab, shown in the figure below, is used to specify the BOD influent element as one of two possible types: **constant** or **variable**. A constant BOD influent's wastewater flow/composition is constant during a steady state simulation, but may be modified by the user during a dynamic simulation. The wastewater flow/composition of a constant BOD influent may be changed during a dynamic simulation only if the simulator is paused.



The BOD influent type tab

A variable BOD influent element represents a user-specified time-varying flow / composition stream entering the system. For steady state calculations the mass average of the time-varying flow / composition pattern is used. Each set of conditions has an associated starting time; this time is the dynamic simulation time at which these conditions will start to be used. The BOD influent places the following restrictions on the starting time for each set of conditions:

1. The starting time for each set of conditions must be greater than the starting time for the previous set.
2. The first set of conditions must start at time 0.
3. The starting time of the final set of conditions must be less than the time specified in the **Cycle duration** edit box.

During a dynamic simulation each set of BOD influent conditions is used from when the simulation time equals its starting time until the simulation time reaches the starting time of the following set of conditions. The final set of BOD influent conditions is used from its starting time until the **Cycle duration** time is reached. At the end of the cycle the process is repeated using the **Cycle duration** time as time zero.

Clicking the **Edit data** button will present you with a dialog that is dependant on the type of BOD influent. If the BOD influent is constant, you will be presented with the **Edit influent** dialog box. If the influent is scheduled, you will be presented with the **Edit influent itinerary** dialog box. For more information on the use of these dialog boxes, please see **Itinerary Editors**.

You also may input data by loading a previously saved data file. Clicking the **From file** button will present you with the **Open influent file** dialog box. Using this dialog box, select the influent data file you want and click **Open** to finish.

Clicking the **Check pH and alkalinity settings** button will check that the pH and alkalinity values that you have entered are consistent. That is, that BioWin is able to provide an acceptable mix of dissolved carbon dioxide, anions and cations to match

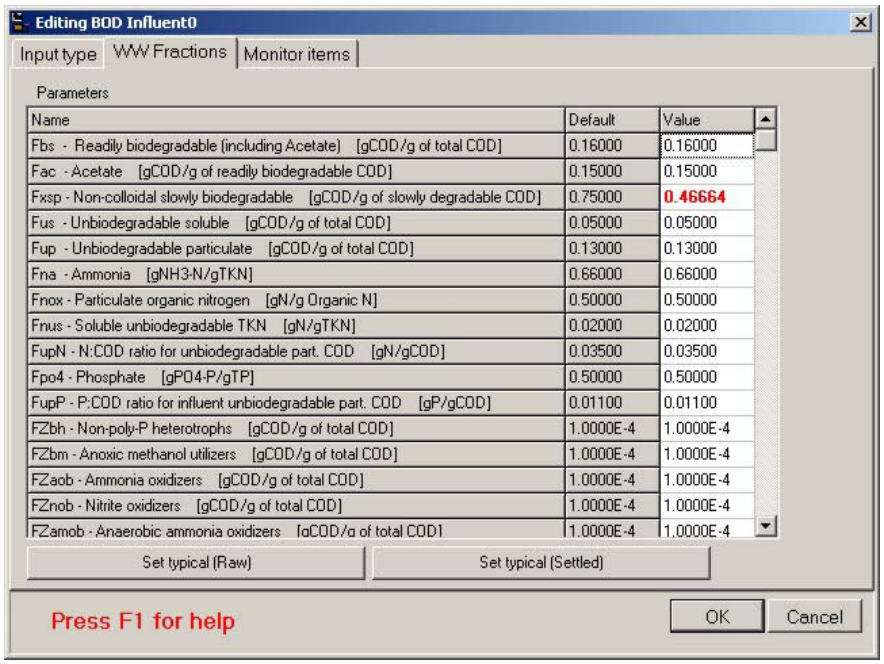
the settings you have specified. This process is recommended although it can be time consuming for large influent itineraries (e.g. greater than one thousand rows). Should you decide not to do it then BioWin will generate an alarm if the condition is encountered during a simulation.

You also may save a data file if you feel that you may want to use it again. Clicking the **To file** button will present you with the **Save influent file** dialog box. Using this dialog box, save the data file under a name and location that you will remember.

Influent (BOD) Wastewater Fractions

The **Wastewater fractions** tab, shown in the figure below, is used to specify the fractional composition of the BOD influent wastewater. Note that these fractions are in terms of COD. Even though the overall wastewater strength is entered in terms of BOD, the BioWin model calculates the strength in terms of COD, and uses these fractions to place values in the state vector.

It should also be noted that the value of the fraction F_{xsp} is calculated by BioWin based on the VSS of the BOD influent. Therefore any value entered by the user for this fraction will be overwritten.



The BOD influent wastewater fractions tab

You may specify the following fractional values:

Name	Description
F_{bs}	Fraction of total influent COD which is readily biodegradable [(S_{bsc} + S_{bsa}) / Total influent COD]
F_{ac}	Fraction of readily biodegradable COD which is VFAs [S_{bsa} / (S_{bsa}+S_{bsc})]
F_{xsp}	Fraction of slowly biodegradable influent COD which is particulate [X_{sp} / (X_{sc} + X_{sp})]. Note that this value is back-calculated by BioWin based on the VSS value entered in the BOD influent.
F_{us}	Fraction of total influent COD which is soluble

	unbiodegradable
F_{up}	Fraction of total influent COD which is particulate unbiodegradable
F_{na}	Fraction of influent TKN which is ammonia
F_{nox}	Fraction of influent biodegradable organic nitrogen which is particulate
F_{nus}	Fraction of influent TKN which is soluble unbiodegradable
F_{po4}	Fraction of influent TP which is phosphate
F_{upN}	The N:COD ratio for the influent particulate unbiodegradable COD
F_{upP}	The P:COD ratio for the influent particulate unbiodegradable COD
FZ_{bh}	Fraction of total influent COD which is non-polyP heterotrophic organisms.
FZ_{aob}	Fraction of total influent COD which is ammonia oxidizing organisms.
FZ_{nob}	Fraction of total influent COD which is nitrite oxidizing organisms.
FZ_{amob}	Fraction of total influent COD which is anaerobic ammonia oxidizing organisms.
FZ_{bp}	Fraction of total influent COD which is polyP heterotrophic organisms.
FZ_{bpa}	Fraction of total influent COD which is propionic acid acetogen organisms.
FZ_{bam}	Fraction of total influent COD which is acetoclastic methanogen organisms.
FZ_{bhm}	Fraction of total influent COD which is H_2-utilizing methanogen organisms
FZ_{bm}	Fraction of total influent COD which is anoxic methanol utilizing organisms

Note: If you enter values for FZ_{bh}, FZ_{aob}, FZ_{nob}, FZ_{amob}, FZ_{ba}, FZ_{bp}, FZ_{bpa}, Fz_{bam}, FZ_{bhm}, or FZ_{bm} remember that organisms contain nitrogen and phosphorus.

An explanation of the fractionation of influent nitrogen may be helpful at this point. Ammonia is given by:

$$NH_3 = F_{na} \cdot TKN_t$$

Soluble unbiodegradable organic nitrogen is given by:

$$N_{US} = F_{nu} \cdot TKN_t$$

Nitrogen from organisms present in the influent is calculated by the sum of the products of the various organism concentrations and their respective nitrogen fractions, i.e.:

$$OrganismsN = \sum Zb\chi \cdot f_{N,ZB\chi}$$

Unbiodegradable particulate nitrogen is given by:

$$X_{IN} = F_{up,N} \cdot F_{up} \cdot COD_t$$

The remaining organic nitrogen is broken into particulate and soluble components. Particulate biodegradable organic nitrogen is given by:

$$X_{ON} = (TKN_t - NH_3 - N_{US} - X_{IN} - OrganismsN) \cdot F_{nox}$$

Soluble biodegradable organic nitrogen is given by:

$$N_{OS} = (TKN_t - NH_3 - N_{US} - X_{IN} - OrganismsN) \cdot (1 - F_{nox})$$

Similarly, an explanation of the fractionation of influent phosphorus is as follows. Soluble orthophosphate is given by:

$$PO_4 = F_{po4} \cdot TP$$

Phosphorus from organisms present in the influent is calculated by the sum of the products of the various organism concentrations and their respective phosphorus fractions, i.e.:

$$OrganismsP = \sum Zb\chi \cdot f_{P,ZB\chi}$$

Unbiodegradable particulate phosphorus is given by:

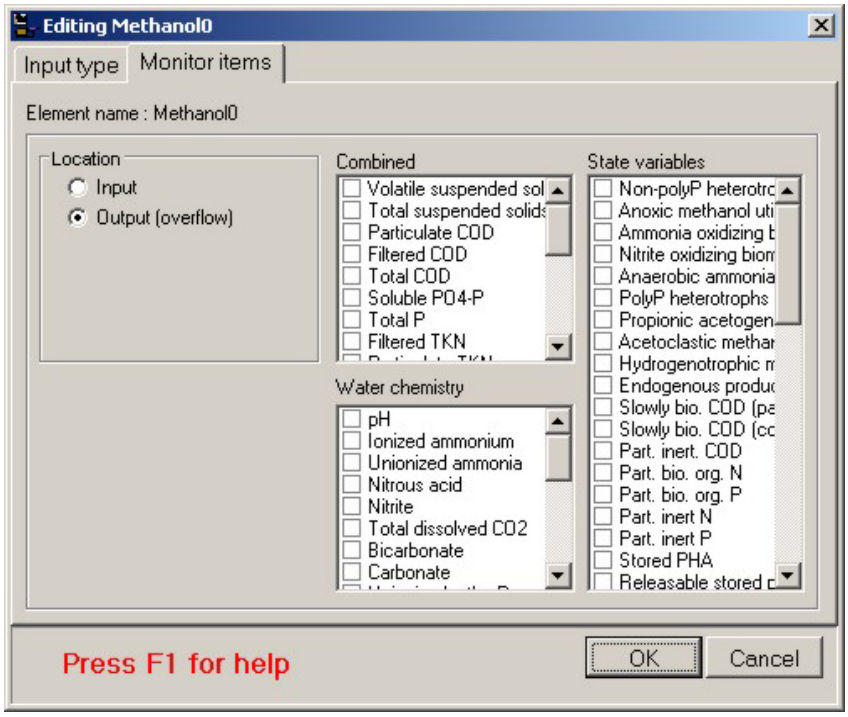
$$X_{IP} = F_{up,P} \cdot F_{up} \cdot COD_t$$

The remaining particulate biodegradable organic phosphorus is given by:

$$X_{OP} = TP - PO_4 - X_{IP} - OrganismsP$$

Methanol

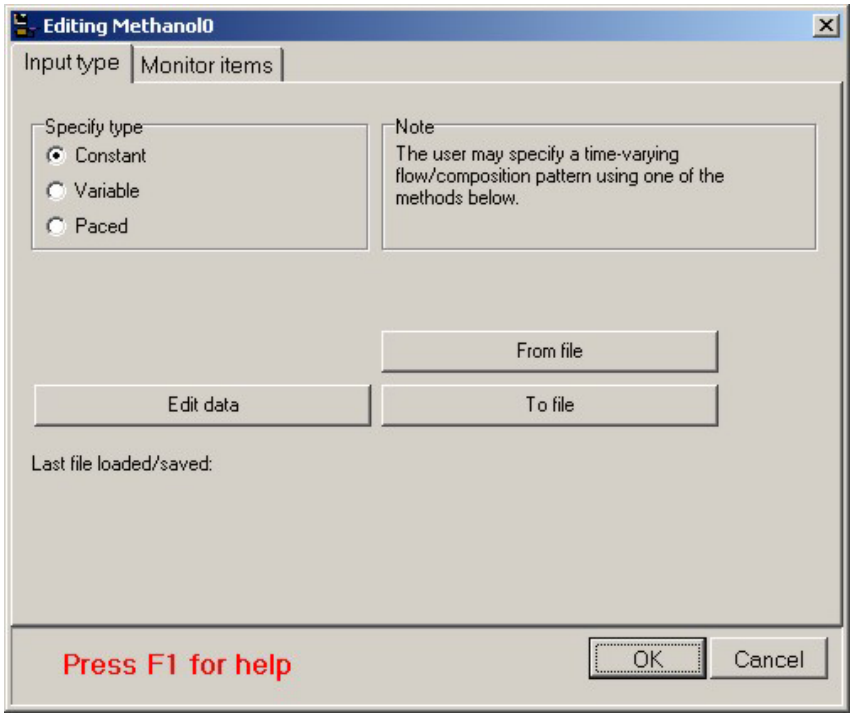
The methanol influent element is used to control the flow/composition of a methanol addition stream to your process configuration. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The methanol influent element monitor items tab

Methanol Influent Type

The **Methanol input type** tab, shown in the figure below, is used to specify the methanol influent element as one of three possible types: **constant** , **variable** or **paced**. A constant methanol influent's flow/composition is constant during a steady state simulation, but may be modified by the user during a dynamic simulation. The methanol flow/composition may be changed during a dynamic simulation only if the simulator is paused.



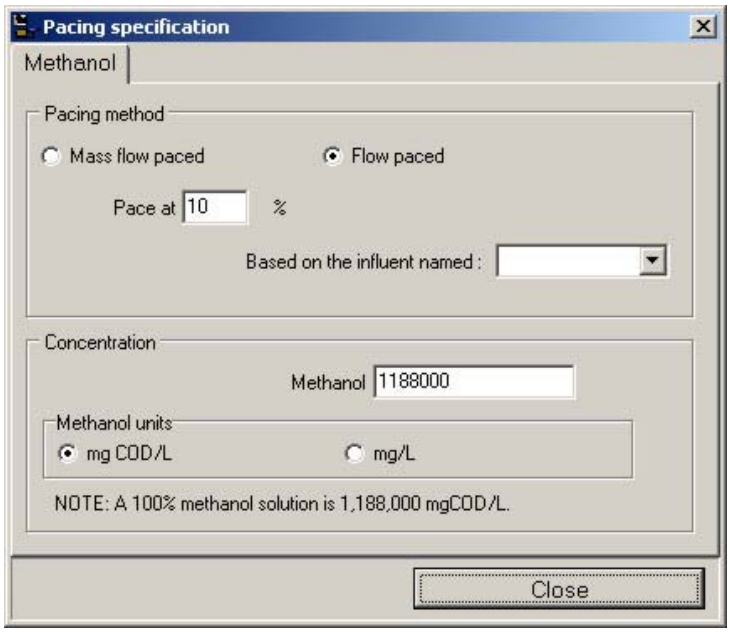
The methanol influent type tab

A methanol influent element represents a user-specified time-varying flow / composition stream entering the system. For steady state calculations the mass average of the time-varying flow / composition pattern is used. Each set of conditions has an associated starting time; this time is the dynamic simulation time at which these conditions will start to be used. The methanol influent places the following restrictions on the starting time for each set of conditions:

1. The starting time for each set of conditions must be greater than the starting time for the previous set.
2. The first set of conditions must start at time 0.
3. The starting time of the final set of conditions must be less than the time specified in the **Cycle duration** edit box.

During a dynamic simulation each set of methanol influent conditions is used from when the simulation time equals its starting time until the simulation time reaches the starting time of the following set of conditions. The final set of methanol influent conditions is used from its starting time until the **Cycle duration** time is reached. At the end of the cycle the process is repeated using the **Cycle duration** time as time zero.

Clicking the **Edit data** button will present you with a dialog that is dependant on the type of methanol influent. If the methanol influent is constant, you will be presented with the **Edit influent** dialog box. If the methanol influent is scheduled, you will be presented with the **Edit influent itinerary** dialog box. For more information on the use of these dialog boxes, please see **Itinerary Editors**. If the methanol influent is flow paced, you will be presented with the Methanol **Pacing Specification** dialog box.



The methanol pacing specification dialog box

There are two options for pacing method:

1. Mass flow paced
2. Flow paced

If **Mass flow paced** is selected, you may enter a number in the **Pace at** text box area as a **% of the mass of,** and then select an influent component as the basis for flow pacing from the drop list of components. You can also specify the influent element as the basis for mass flow pacing from the drop list of influent elements in the configuration.

If **Flow paced** is selected, you may enter a number in the **Pace at** text box area as a **% ,** and then select the influent element as the basis for flow pacing from the drop list of influent elements in the configuration.

You also may input variable methanol influent data by loading a previously saved data file. Clicking the **From file** button will present you with the **Open influent file** dialog box. Using this dialog box, select the methanol influent data file you want and click **Open** to finish.

You also may save a data file if you feel that you may want to use it again. Clicking the **To file** button will present you with the **Save influent file** dialog box. Using this dialog box, save the data file under a name and location that you will remember.

Specifying Methanol Influent Concentrations

BioWin places default values for a 100 % methanol solution in the methanol influent element. This section outlines how these numbers are calculated. Using the

following methodology, you can specify concentration values for different methanol strengths.

COD / Mass Ratio

The COD / mass ratio for methanol (CH_3OH) can be calculated as follows from balancing the equation for the complete oxidation of methanol to carbon dioxide and water:

$$CH_3OH + \frac{3}{2}O_2 \Rightarrow CO_2 + 2H_2O$$

That is, 1.5 moles (48 g) of oxygen is required to oxidize 1 mole (32 g) of methanol. So the COD / mass ratio for methanol is (48 g / 32 g) 1.5. You can see the methanol concentration number changing by a factor of 1.5 in BioWin when you change between methanol concentration units of mg/L and mgCOD/L.

Methanol Composition

From *Chemical Engineer's Handbook*, the specific gravity of pure methanol is 0.792. That is, it has a density of 0.792 kg/L. Using the COD / mass ratio calculated above, this corresponds to a COD concentration of (1.5*0.792) 1.188 kgCOD/L, or 1, 188, 000 mgCOD/L.

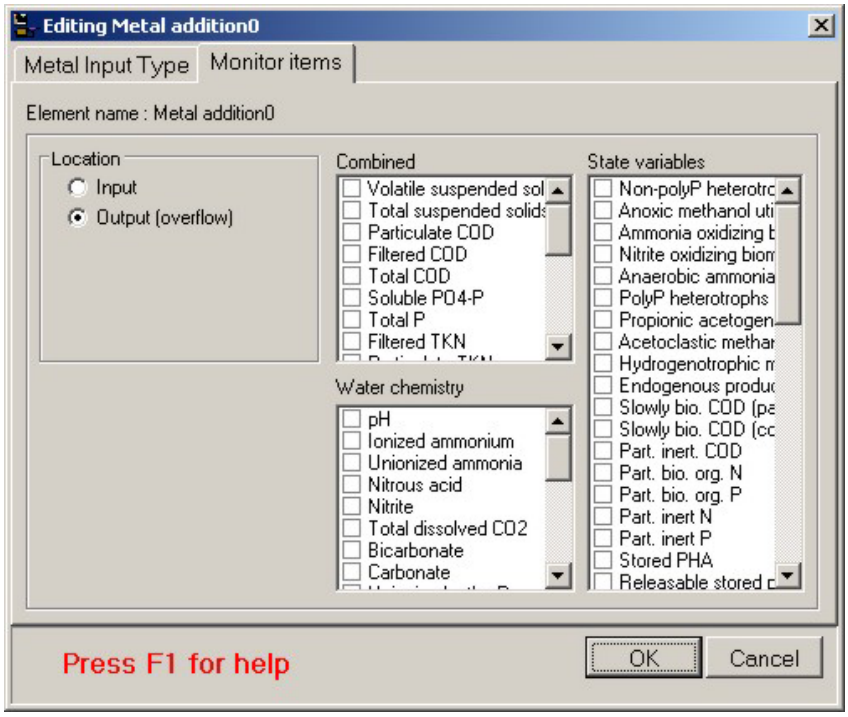
Note that if your methanol solution is specified in terms of molar concentration, convert to a mass concentration using the molecular weight for methanol of 32 g / mole.

Reference

Perry, R.H., Chilton, C.H. (1973) *Chemical Engineer's Handbook.* McGraw-Hill Inc.

Metal Addition Influent

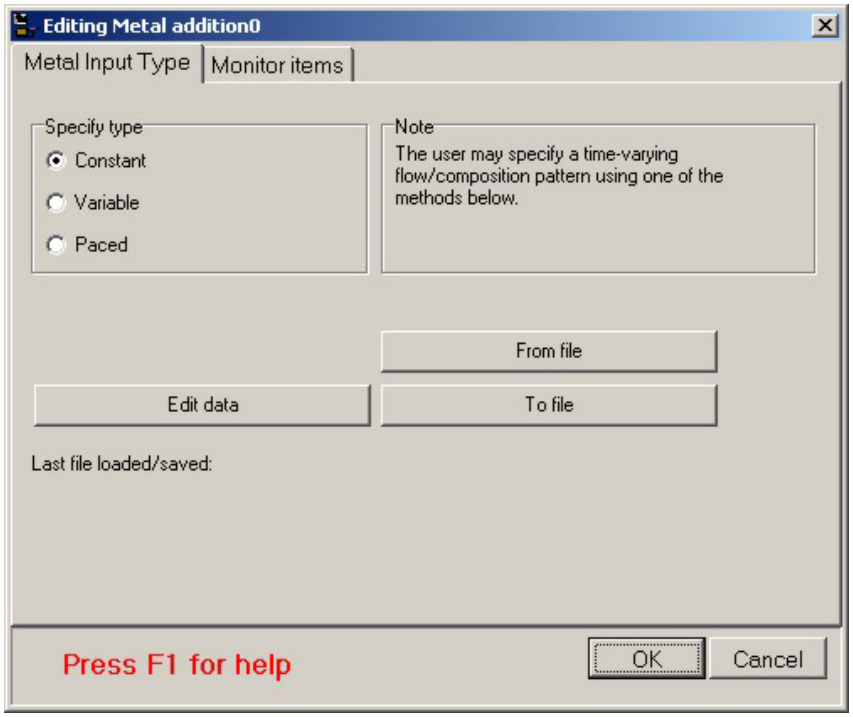
The metal addition influent element is used to control the flow/composition of a metal addition stream to your process configuration. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The metal addition influent element monitor items tab

Metal Addition Influent Type

The **Metal addition input type** tab, shown in the figure below, is used to specify the metal addition influent element as one of two possible types: **constant**, **variable,** or **paced**. A constant metal addition influent's flow/composition is constant during a steady state simulation, but may be modified by the user during a dynamic simulation. The metal flow/composition may be changed during a dynamic simulation only if the simulator is paused.



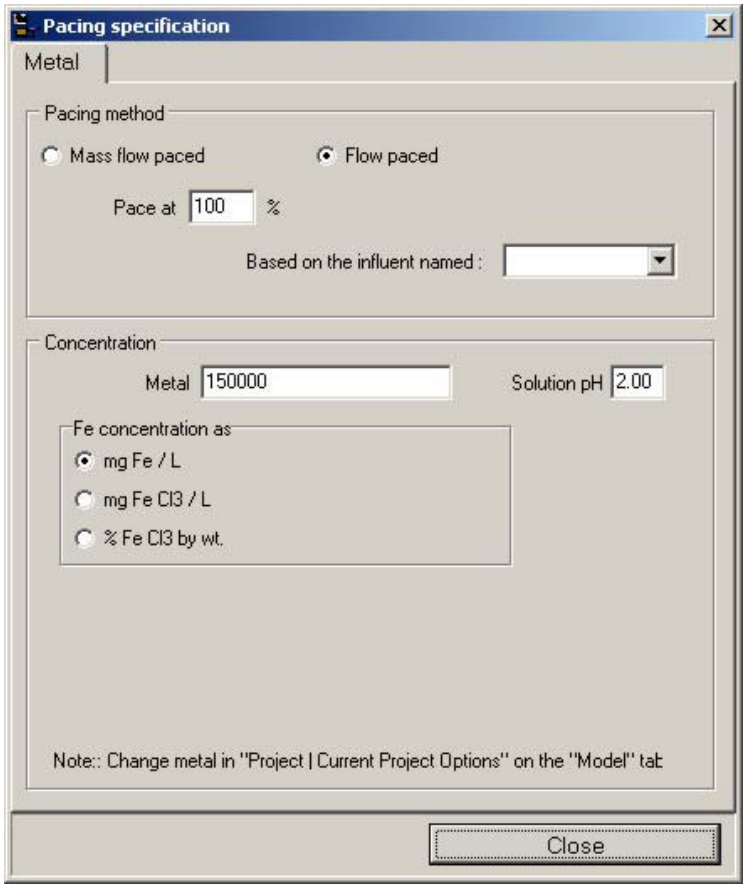
The metal addition influent type tab

A metal addition influent element represents a user-specified time-varying flow / composition stream entering the system. For steady state calculations the mass average of the time-varying flow / composition pattern is used. Each set of conditions has an associated starting time; this time is the dynamic simulation time at which these conditions will start to be used. The metal addition influent places the following restrictions on the starting time for each set of conditions:

1. The starting time for each set of conditions must be greater than the starting time for the previous set.
2. The first set of conditions must start at time 0.
3. The starting time of the final set of conditions must be less than the time specified in the **Cycle duration** edit box.

During a dynamic simulation each set of metal addition influent conditions is used from when the simulation time equals its starting time until the simulation time reaches the starting time of the following set of conditions. The final set of metal addition influent conditions is used from its starting time until the **Cycle duration** time is reached. At the end of the cycle the process is repeated using the **Cycle duration** time as time zero.

Clicking the **Edit data** button will present you with a dialog that is dependant on the type of metal addition influent. If the metal addition influent is constant, you will be presented with the **Edit influent** dialog box. If the metal addition influent is scheduled, you will be presented with the **Edit influent itinerary** dialog box. For more information on the use of these dialog boxes, please see **Itinerary Editors**. . If the metal addition influent is flow paced, you will be presented with the Metal **Pacing Specification** dialog box.



The metal pacing specification dialog box

There are two options for pacing method:

1. Mass flow paced
2. Flow paced

If **Mass flow paced** is selected, you may enter a number in the **Pace at** text box area as a **% of the mass of,** and then select an influent component as the basis for flow pacing from the drop list of components. You can also specify the influent element as the basis for mass flow pacing from the drop list of influent elements in the configuration.

If **Flow paced** is selected, you may enter a number in the **Pace at** text box area as a **% ,** and then select the influent element as the basis for flow pacing from the drop list of influent elements in the configuration.

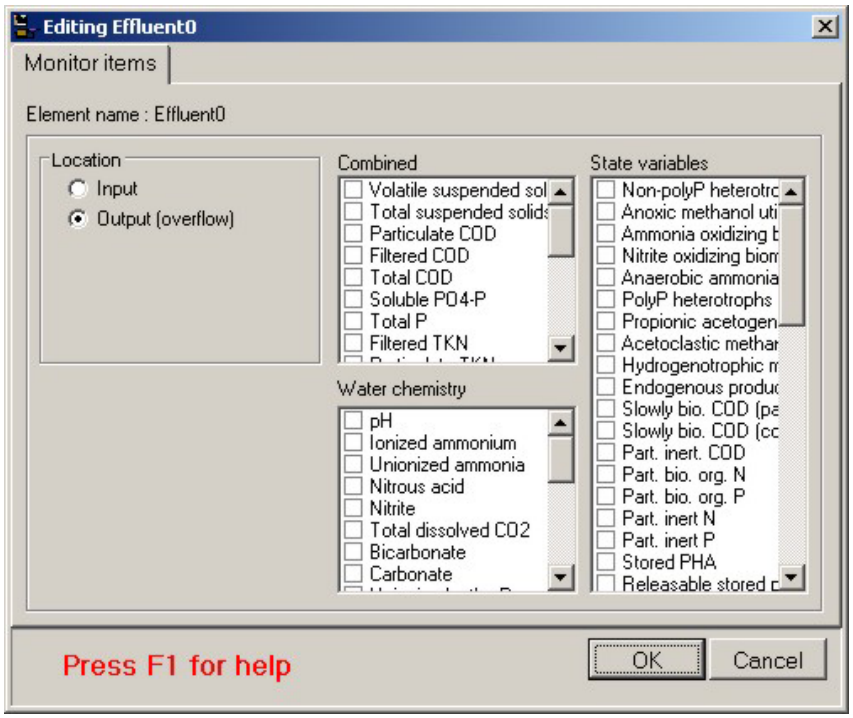
You also may input data by loading a previously saved data file. Clicking the **From file** button will present you with the **Open influent file** dialog box. Using this dialog box, select the metal addition influent data file (**.mef*) you want and click **Open** to finish.

You also may save a data file if you feel that you may want to use it again. Clicking the **To file** button will present you with the **Save influent file** dialog box. Using this dialog box, save the data file under a name and location that you will remember.

Note: You can change the metal used by the metal addition influent on the **Model** tab which can be accessed via the **Project|Current Project Options** menu command.

Effluent

The effluent element has no model. It is used as a receptacle for the conduit from the final element in a chain. As such the effluent element is used primarily for the various checks that BioWin performs on the system integrity. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.

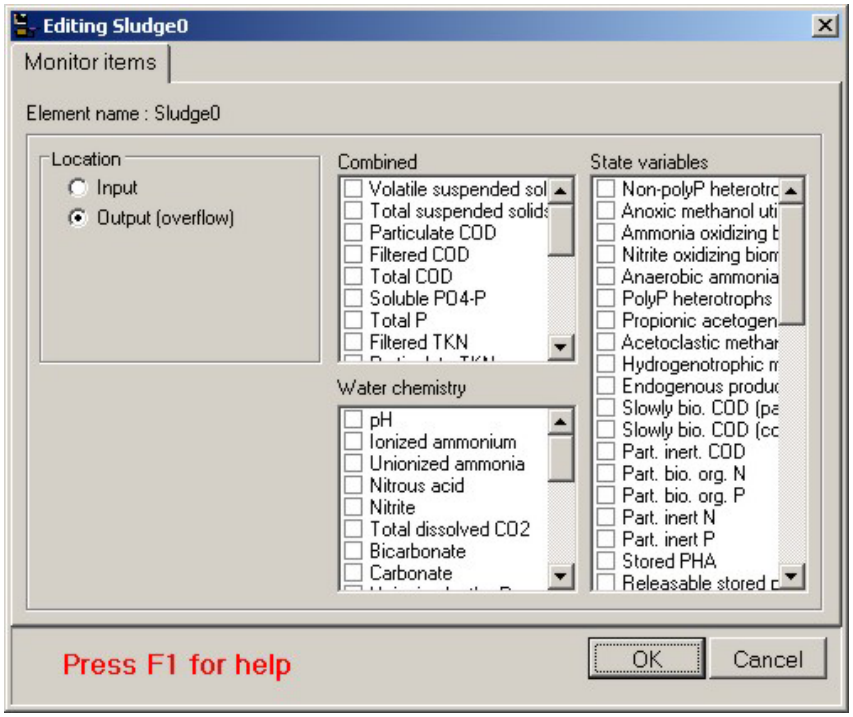


The effluent monitor items tab

Sludge Effluent

The sludge effluent element has no model. It is used as a receptacle for the conduit from the final element in a chain. As such the effluent element is used primarily for the various checks that BioWin performs on the system integrity. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.

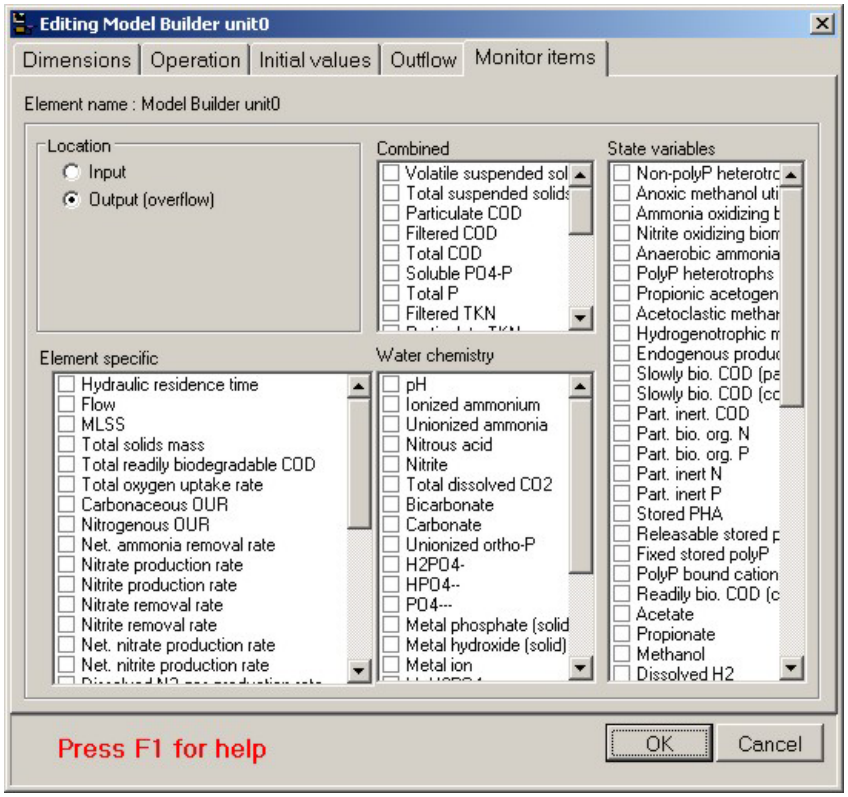
The sludge effluent element differs from a regular effluent element only in the information that is displayed in the Main Simulator Window Summary Panes when you hover over it with the cursor. Information displayed includes VSS and TSS expressed in % solids, nutrient percentages of the sludge, and sludge mass rates. Therefore, this element is useful as a receptacle for WAS flows and other solids train process flows.



The sludge effluent monitor items tab

Model Builder Reactor

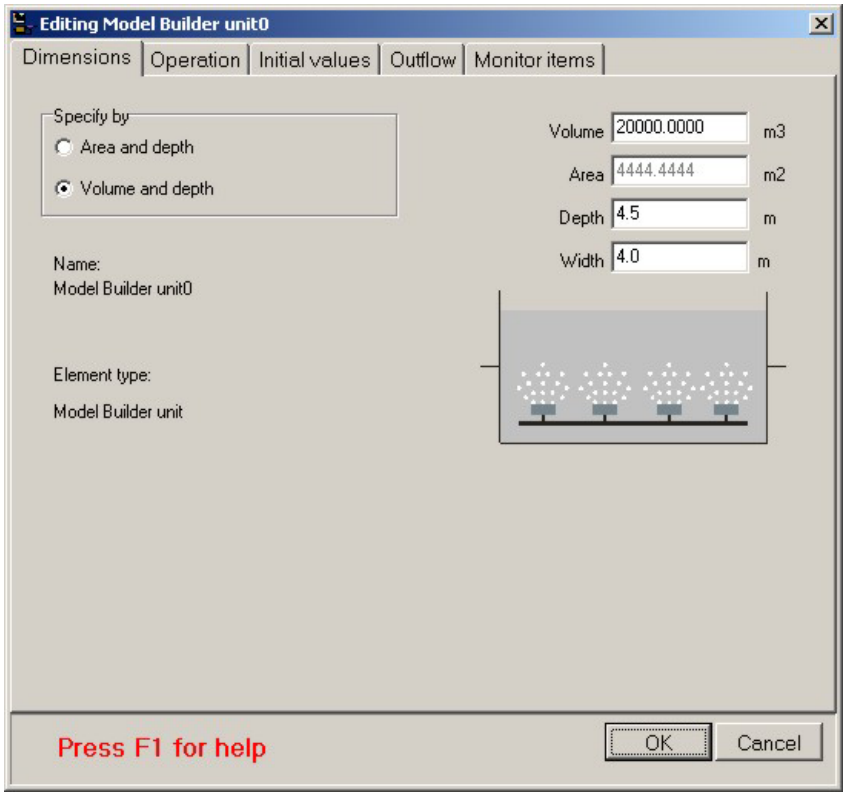
The model builder reactor element allows you to incorporate your own models into BioWin simulations. This element allows you to input your own stoichiometry and rate equations involving any or all of BioWin's state variables. You may specify parameters related to the operation and control of the model builder reactor element, the volume variability, and the model that it uses. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The model builder element monitor items tab

Model Builder Reactor Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of a model builder reactor element.

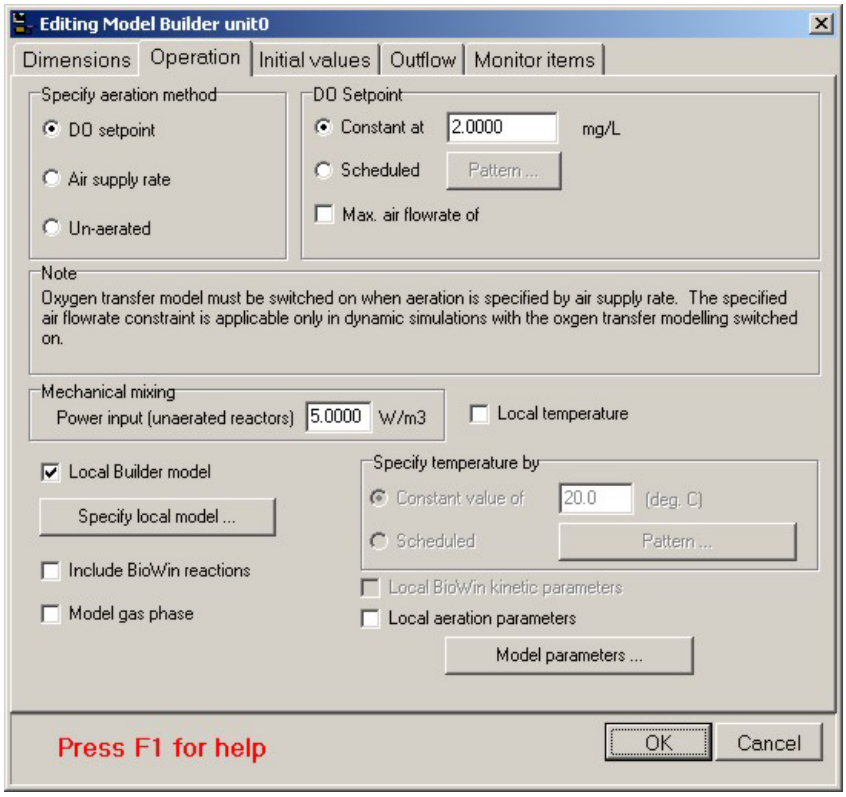


The model builder reactor dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the model builder reactor element area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the model builder reactor element volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a Width for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

Model Builder Reactor Operation

The **Operation** tab, shown below, allows the user to enter operating parameters for a model builder reactor element such as aeration specifications and local temperature, as well as specify user-defined model stoichiometry and rate equations.



The model builder reactor operation tab

There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **Edit DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **Edit air flow itinerary** dialog box).

You may also specify that the model builder reactor element is **Unaerated**.

The mechanical mixing **Power input (unaerated reactors)** can be specified. This is used to calculate the velocity gradient in the vessel.

A **Local temperature** also may be specified for a model builder reactor element. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

There are a number of methods for specifying what model the model builder reactor element uses. If you want to have a local, user-defined model where you set up stoichiometry and rate equations, select the box labeled **Local Builder model**. This will activate the button labeled **Specify local model…**. Clicking this button

will open the **Model Builder** dialog box, which allows you to enter stoichiometry and rate equations for any or all of BioWin's state variables. The use of this tool is explained in more detail in the **Model Builder** section.

If you wish, you may also choose to incorporate BioWin reactions into your model builder reactor element. If you want to do this, select the box labeled **Include BioWin reactions**.

Note that if you wish to change any of the aeration model parameters, click the **Model parameters…** button to open the **Model parameter editor** dialog box (Note that this changes the aeration model parameters on a local level only).

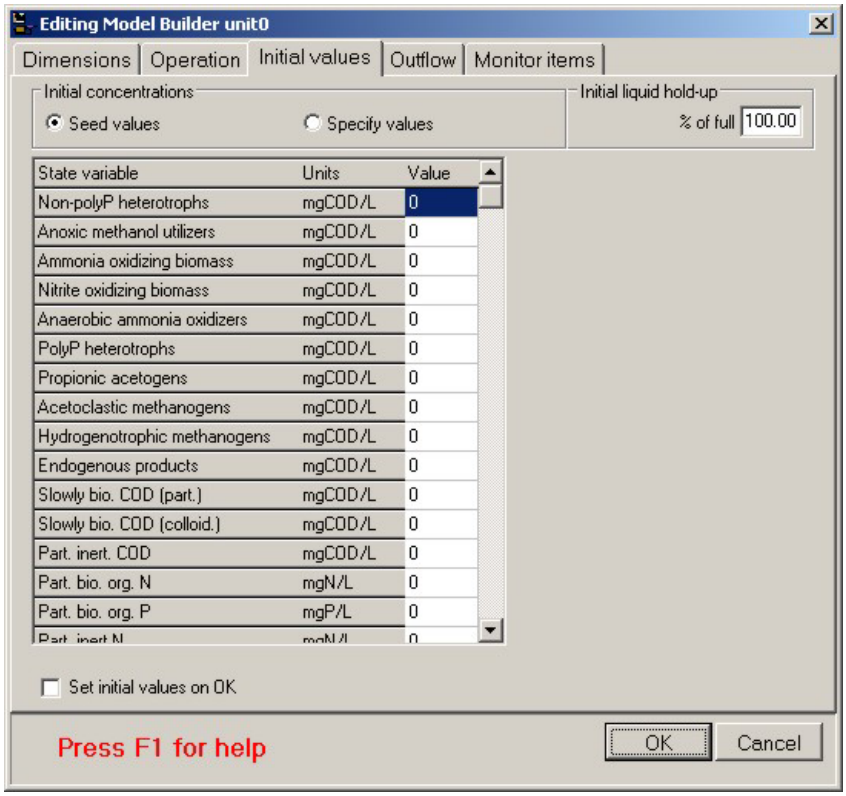
You also can specify **Local BioWin kinetic parameters** for the activated sludge model used in the model builder reactor element, if you have selected the **Include BioWin reactions** option. If you click on the check box for local BioWin kinetic parameters, then clicking the **Model parameters…** button opens the **Model parameter editor** dialog box allowing you access to the various activated sludge model kinetic parameters.

To specify local aeration parameters (e.g. alpha and beta factors), check the box labeled **Local aeration parameters**.

To model the concentrations of the constituents of the aeration gas (e.g. oxygen, carbon dioxide, etc.), check the box labeled **Model gas phase**. If this option is selected you may also enter a percentage value for the **Gas hold-up** (i.e. the percentage of the total reactor volume occupied by aeration gas).

Model Builder Reactor Initial Values

The **Initial values** tab, shown below, is used for specifying the initial settings for model builder reactor element concentrations and volume.

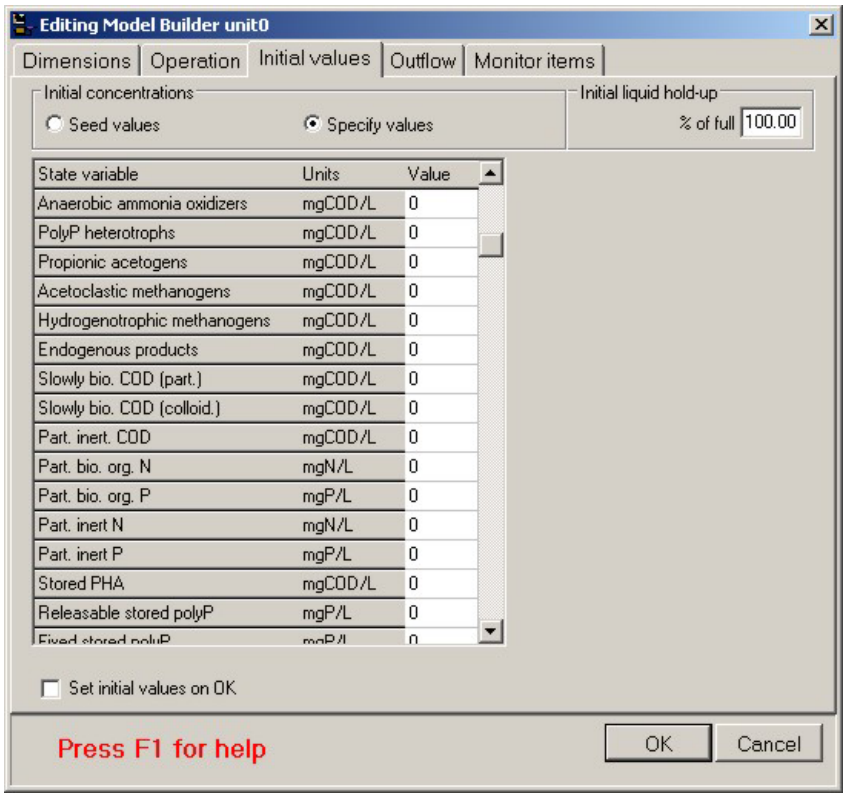


The model builder reactor element initial values tab with the Seed values *option active*

Use the **Initial liquid hold-up** field to enter a value to specify the **% of full** setting. This initial liquid hold-up volume will be used for steady state calculations. It should be noted here that the lower and upper limits on this value are 0.02 and 99.98 %, respectively. This reflects the fact that this is the liquid volume, and allows for small differences between *liquid* volume and *reactor* volume for inlet and outlet pipes.

There are two methods for setting up **Initial concentrations** in the model builder reactor element. The dialog box shown above illustrates the first case where the **Seed values** option is selected. When this option is selected, BioWin applies default seed values for all the state variables (except volume, which you specify) in the same manner it would for bioreactor elements.

The dialog box shown below illustrates the second case where the **Specify values** option is selected. In this case, an editable list of state variables is displayed in the dialog box. This allows you to enter your own seed values for state variables in the model builder reactor element. These will be inserted in the model builder reactor element when you begin a steady state simulation, or when you begin a dynamic simulation and choose to use seed values.



The model builder reactor element initial values tab with the Specify values *option active*

When the specified seed values get inserted in the model builder reactor element is determined by the **Set initial values on OK** option. If this option **is not** checked:

- The specified seed values will be placed in the model builder reactor element at the beginning of a steady state simulation if you elect to start from seed conditions;

- The specified values will be placed in the model builder reactor element at the beginning of a dynamic simulation **if** you elect to seed the simulation.

If this option **is** checked, the specified values are placed in the model builder reactor element when you click **OK** to close the dialog box, **overriding any existing state variable values**. Therefore, the state variable values at the beginning of a dynamic simulation will be the same regardless of whether or not you choose to re-seed.

The use of this option is illustrated in the example at the end of the **Variable Volume/Batch Bioreactor Initial Values** section later in this chapter.

Model Builder Reactor Outflow

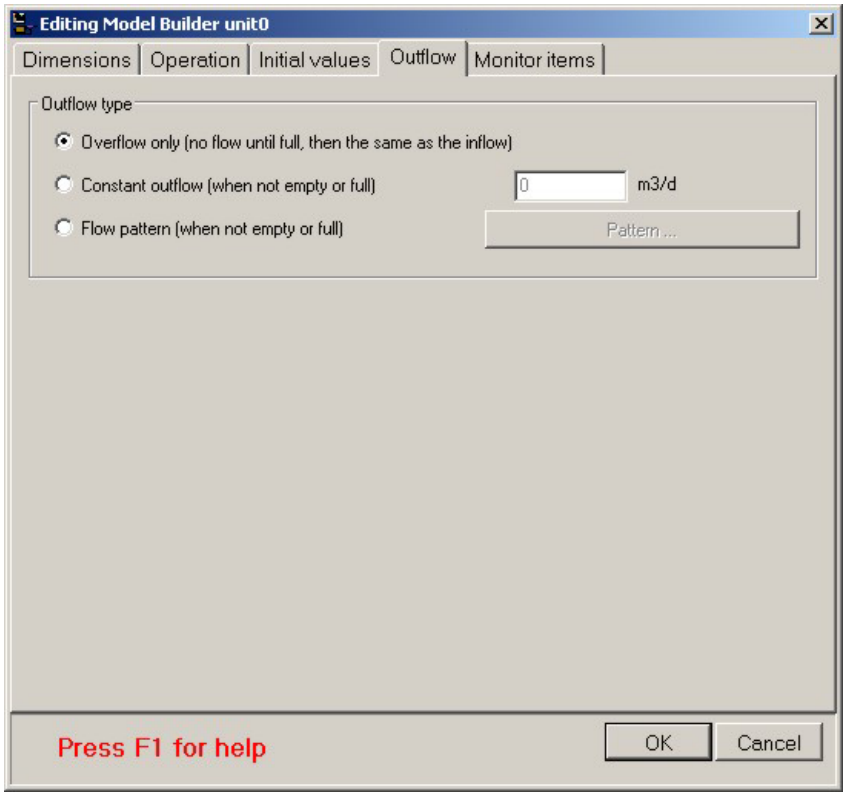
Remember for steady state simulations, the initial liquid hold-up volume will be used for calculations.

The model builder reactor element **Outflow** tab, shown below, is used to specify the overflow behavior. The overflow behavior can be generalized as follows:

- Whenever the model builder reactor element is full, it overflows at the influent rate, regardless of the overflow setting.
- Whenever the model builder reactor element is empty and the outflow rate is set higher than the influent rate, the model builder reactor element will only have an outflow equal to the influent flow, so as not to have negative volume. If the outflow rate is set lower than the influent rate, then the model builder reactor element will begin to fill up.

The following is a description of the behavior of the three outflow settings:

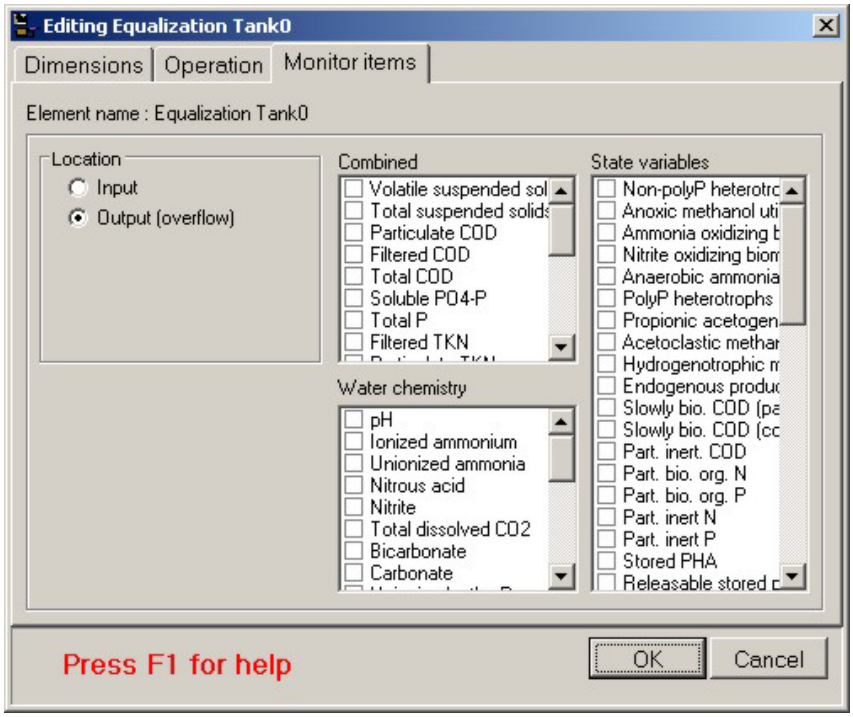
- **Overflow only** – the model builder reactor element fills up, and then overflows at the influent flow rate (this setting could be used to simulate start-up of a bioreactor).
- **Constant outflow** – the outflow always tries to attain the specified constant rate, except when physically constrained (i.e. when the model builder reactor element is full or empty)
- **Flow pattern** – the outflow always tries to attain the current specified pattern rate, except when physically constrained (i.e. when the model builder reactor element is full or empty). To specify a pattern, click the **Pattern…** button when it becomes active. For more information , see the **Liquid outflow itinerary** section.



The model builder reactor element outflow tab

Equalization Tank

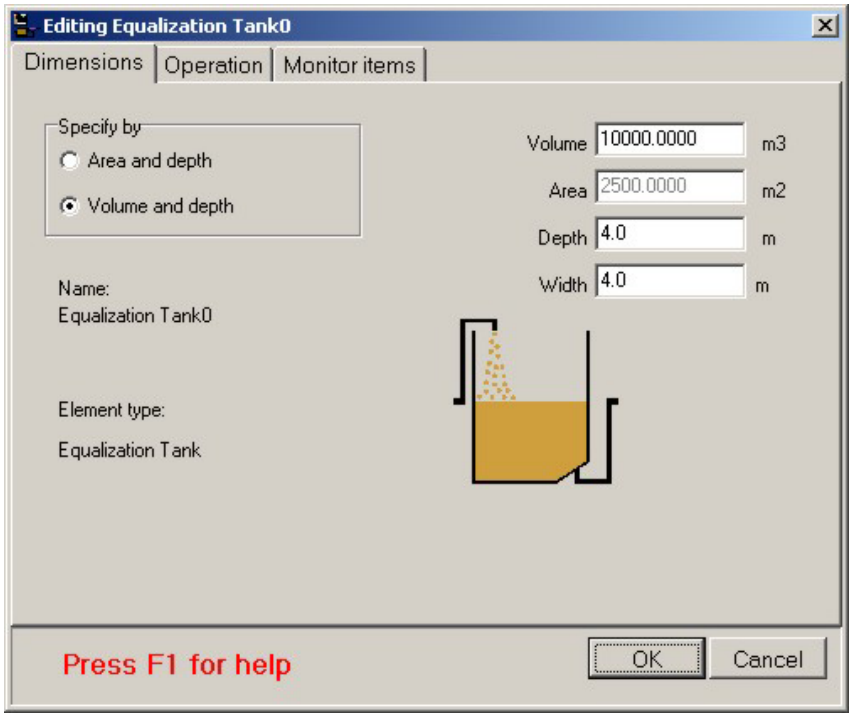
The equalization tank is a completely mixed vessel in which there is no reaction. You may change aspects of the equalization tank's dimensions and operation. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The equalization tank monitor items tab

Equalization Tank Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of a tank element.

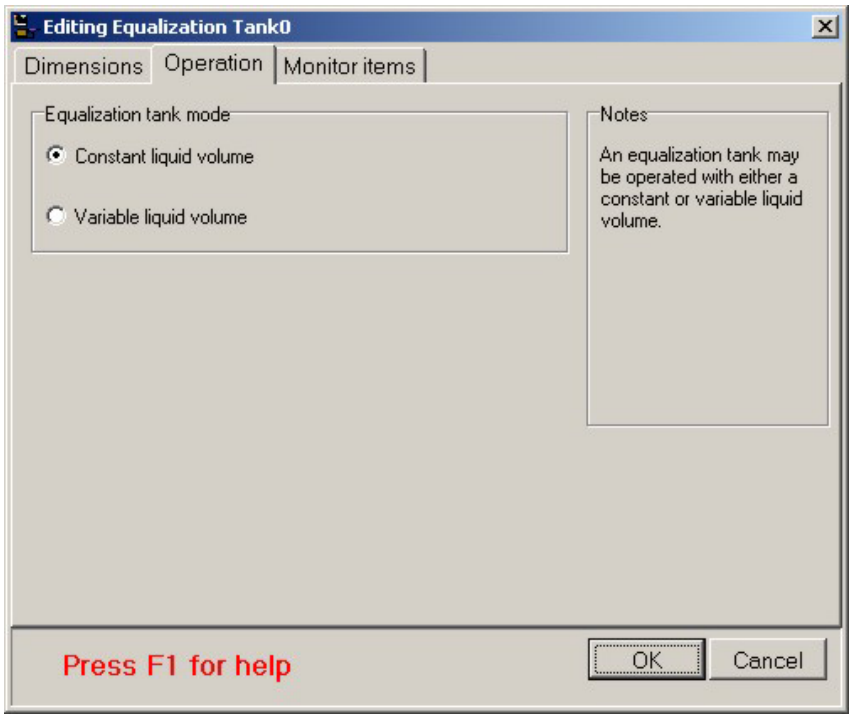


The equalization tank dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the tank area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the tank volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a **Width** for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

Equalization Tank Operation

The **Operation** tab, shown below, allows the user to enter operating parameters for an equalization tank element.



The equalization tank operation tab

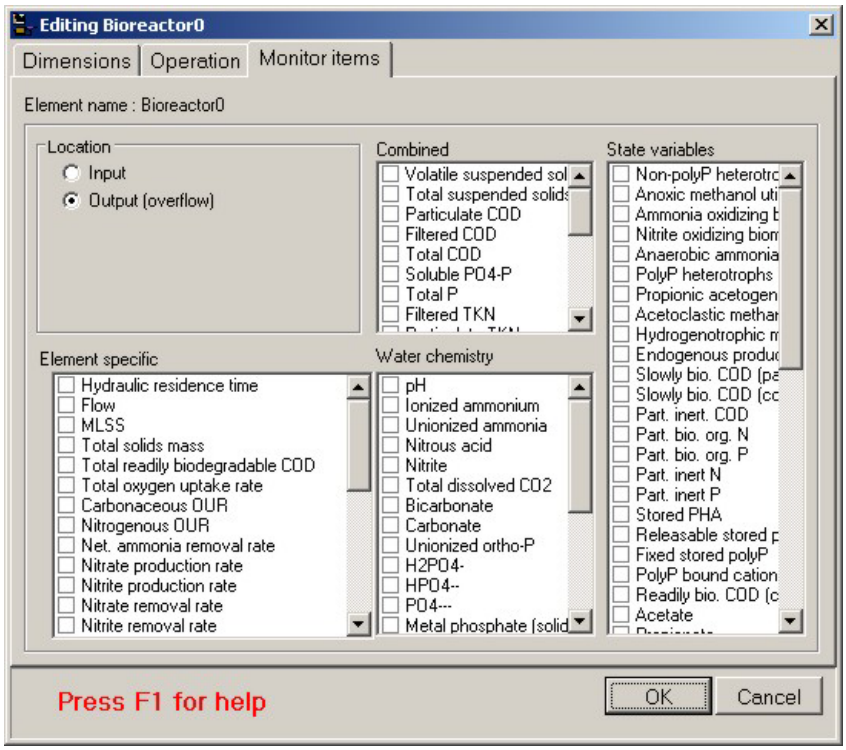
You may specify **Equalization tank mode** (either a constant or variable liquid volume) by clicking on the appropriate radio button. If the **Constant liquid volume** mode is selected, the outflow rate is equal to the inflow rate, and no other operating data are required; the liquid volume is equal to the volume entered in the dimensions tab.

If the **Variable liquid volume** mode is selected, the **Outflow rate** and **Initial liquid hold-up** (as a percentage of the full volume) must be specified. Edit boxes are provided for entering the outflow rate and initial liquid hold-up; flow units are shown adjacent to these.

Bioreactor

The bioreactor simulates the activated sludge process in a continuous stirred tank reactor (CSTR). Complex activated sludge systems can be configured by arranging a number of bioreactors in series/parallel. You may specify parameters related to the

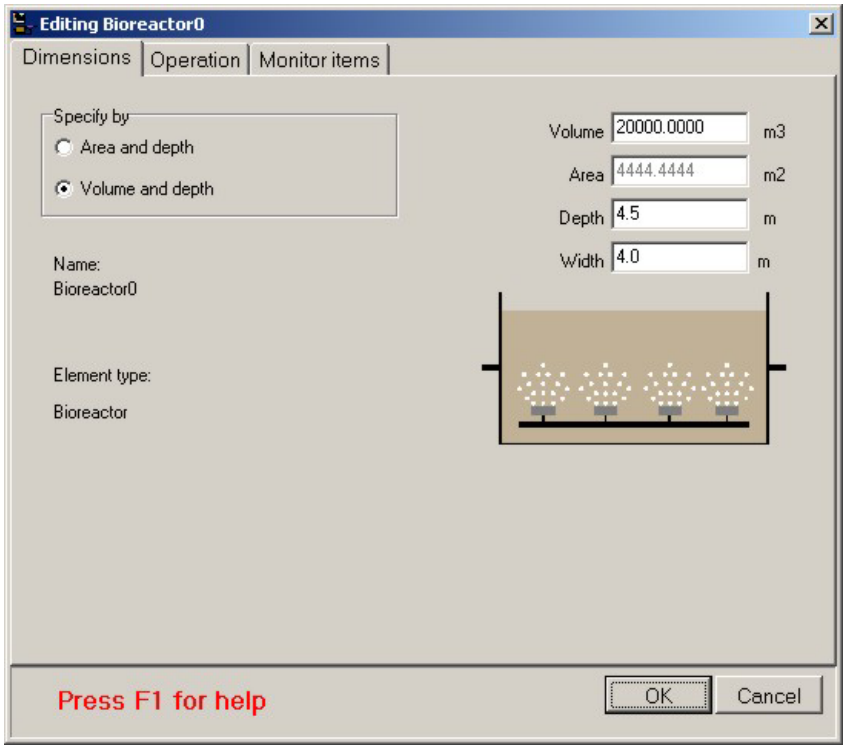
operation and control of the bioreactor element. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The bioreactor monitor items tab

Bioreactor Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of a bioreactor element.

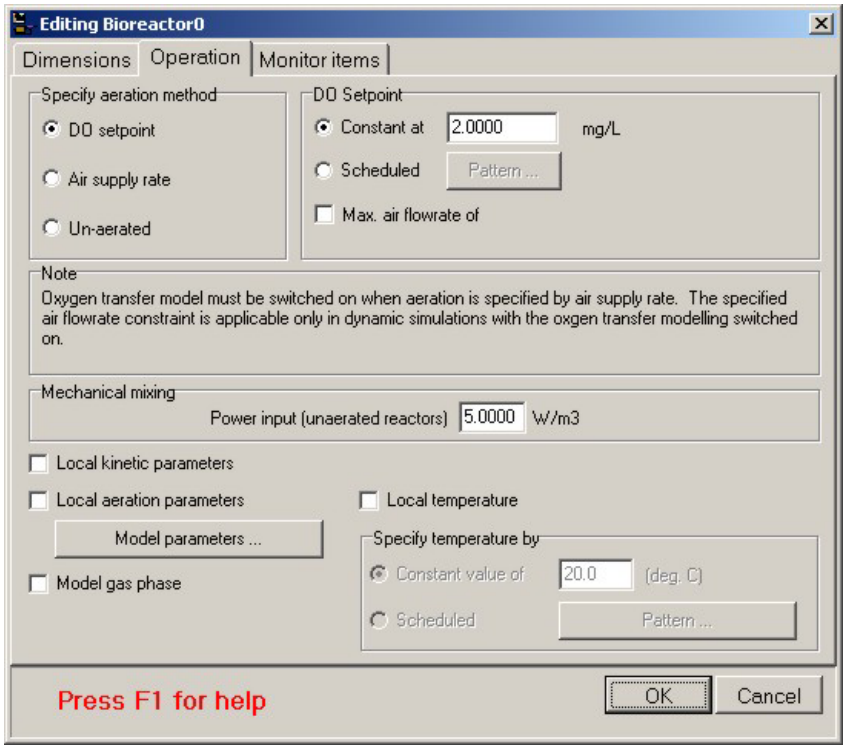


The bioreactor dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the bioreactor area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the bioreactor volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a **Width** for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

Bioreactor Operation

The **Operation** tab, shown below, allows the user to enter operating parameters for a bioreactor element, such as aeration specifications and local temperature, as well as change local model parameters.



The bioreactor operation tab

There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **Edit DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **Edit air flow itinerary** dialog box).

The mechanical mixing **Power input (unaerated reactors)** can be specified. This is used to calculate the velocity gradient in the vessel.

Note : If you wish to change any of the aeration model parameters, click the **Model parameters…** button to open the **Model parameter editor** dialog box (Note that this changes the aeration model parameters on a local level only).

A **Local temperature** also may be specified for a bioreactor element. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

You also can specify **Local kinetic parameters** for the activated sludge model used in the bioreactor element. If you click on the check box for local kinetic parameters, then clicking the **Model parameters…** button opens the **Model**

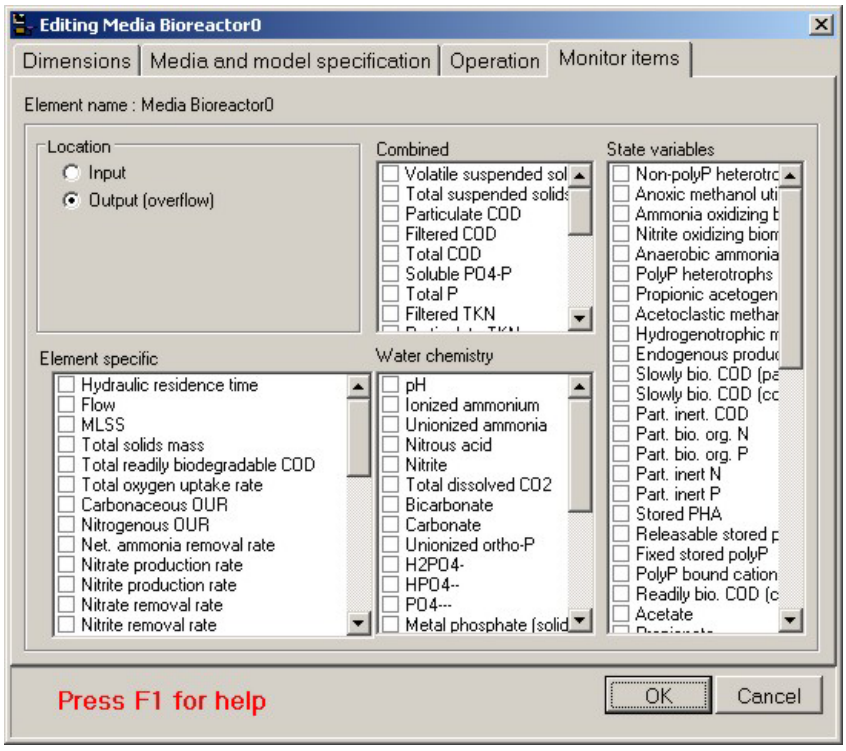
parameter editor dialog box allowing you access to the various activated sludge model kinetic parameters.

To specify local aeration parameters (e.g. alpha and beta factors), check the box labeled **Local aeration parameters**.

To model the concentrations of the constituents of the aeration gas (e.g. oxygen, carbon dioxide, etc.), check the box labeled **Model gas phase**. If this option is selected you may also enter a percentage value for the **Gas hold-up** (i.e. the percentage of the total reactor volume occupied by aeration gas).

Media Bioreactor

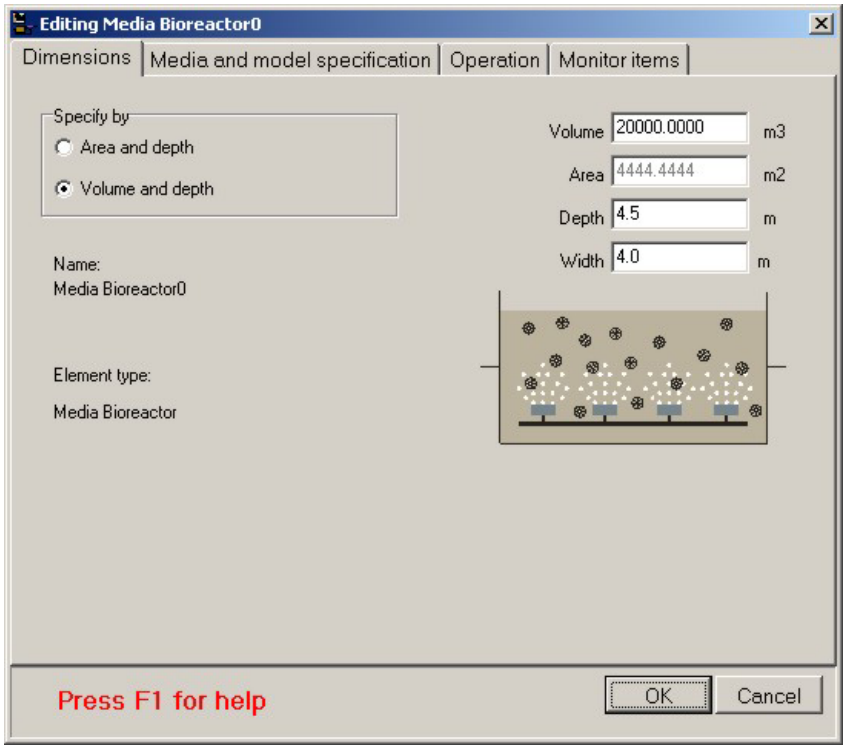
The media bioreactor can be used to simulate activated sludge processes which involve a mixture of fixed and suspended phases. Complex activated sludge systems can be configured by arranging a number of media bioreactors in series/parallel. You may specify parameters related to the operation and control of the media bioreactor element. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The monitor items tab

Media bioreactor Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of a media bioreactor element.

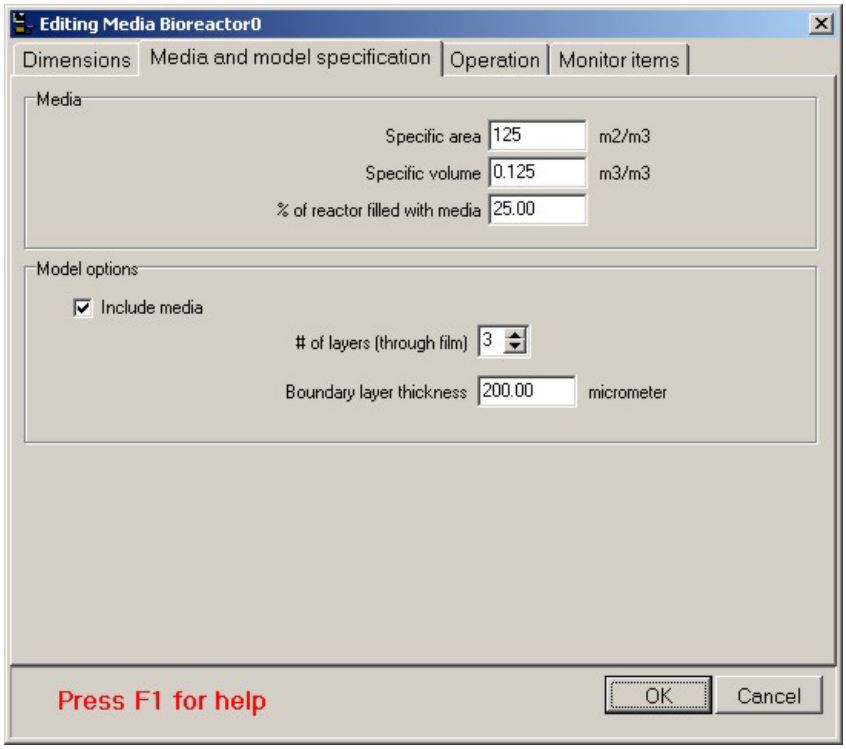


The dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the media bioreactor area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the media bioreactor volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a **Width** for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

Media and model specification

The **Media and model specification** tab, shown below, has two areas. The first area (Model options) allows the user to specify whether the media should be considered for this simulation, the number of layers to use when modeling the biofilm attached to the media and the liquid boundary layer thickness.



The Media and model specification tab

The media area of this tab, allows the user to specify the type and quantity of media to be used in the vessel The "% of the reactor filled with media" determines the amount of media and consequently the area of media for biofilm growth. The media volume is calculated by multiplying the reactor volume with the media fill fraction (the percentage/100) and the "Specific volume". The area for biofilm growth is calculated by multiplying the reactor volume with the media fill fraction (the percentage/100) and the "Specific area". BioWin also attempts to correct the liquid volume for the media and biofilm growth. The liquid volume in the media bioreactor is determined as follows:

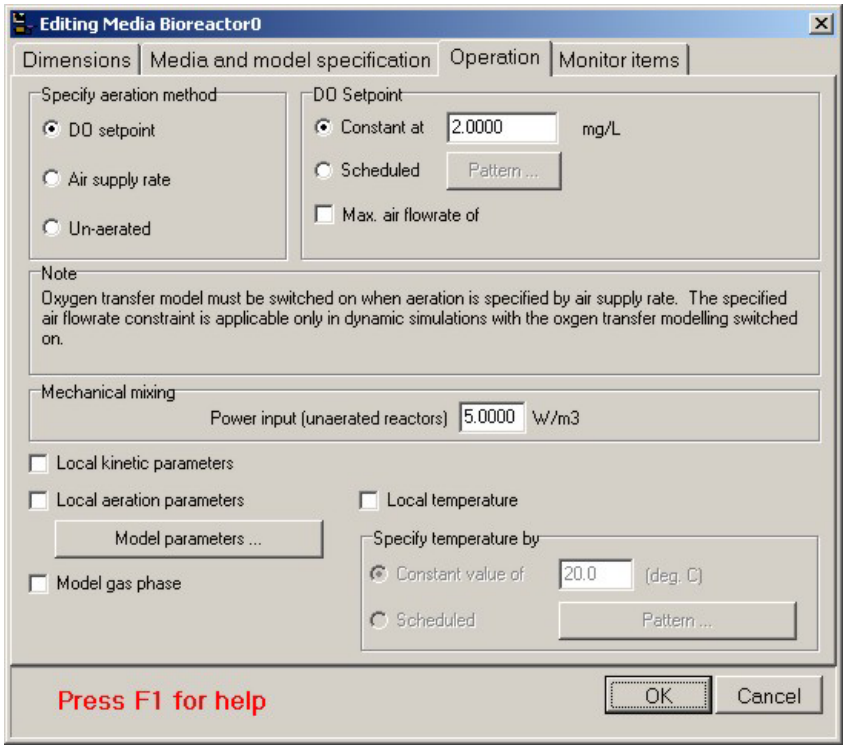
Liquid volume = Vessel volume

– Media Volume

– (Media area * Assumed Film thickness for tank volume correction)

Media bioreactor Operation

The **Operation** tab, shown below, allows the user to enter operating parameters for a media bioreactor element, such as aeration specifications and local temperature, as well as change local model parameters.



The operation tab

There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **Edit DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **Edit air flow itinerary** dialog box).

The mechanical mixing **Power input (unaerated reactors)** can be specified. This is used to calculate the velocity gradient in the vessel.

Note : If you wish to change any of the aeration model parameters, click the **Model parameters…** button to open the **Model parameter editor** dialog box (Note that this changes the aeration model parameters on a local level only).

A **Local temperature** also may be specified for a media bioreactor element. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

You also can specify **Local kinetic parameters** for the activated sludge model used in the media bioreactor element. If you click on the check box for local kinetic

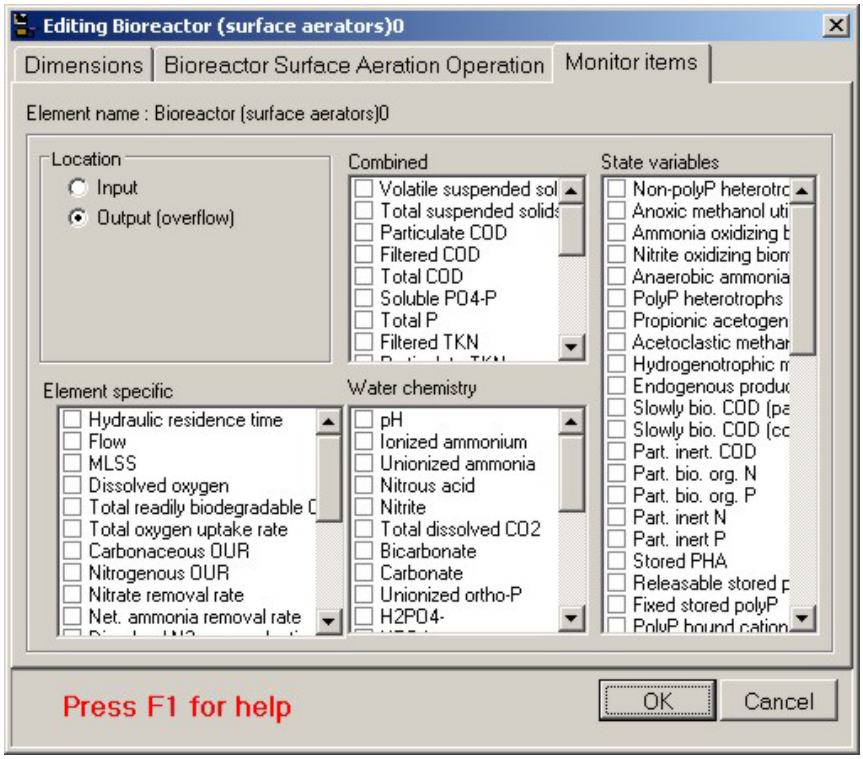
parameters, then clicking the **Model parameters…** button opens the **Model parameter editor** dialog box allowing you access to the various activated sludge model kinetic parameters.

To specify local aeration parameters (e.g. alpha and beta factors), check the box labeled **Local aeration parameters**.

To model the concentrations of the constituents of the aeration gas (e.g. oxygen, carbon dioxide, etc.), check the box labeled **Model gas phase**. If this option is selected you may also enter a percentage value for the **Gas hold-up** (i.e. the percentage of the total reactor volume occupied by aeration gas).

Surface Aerator Bioreactor

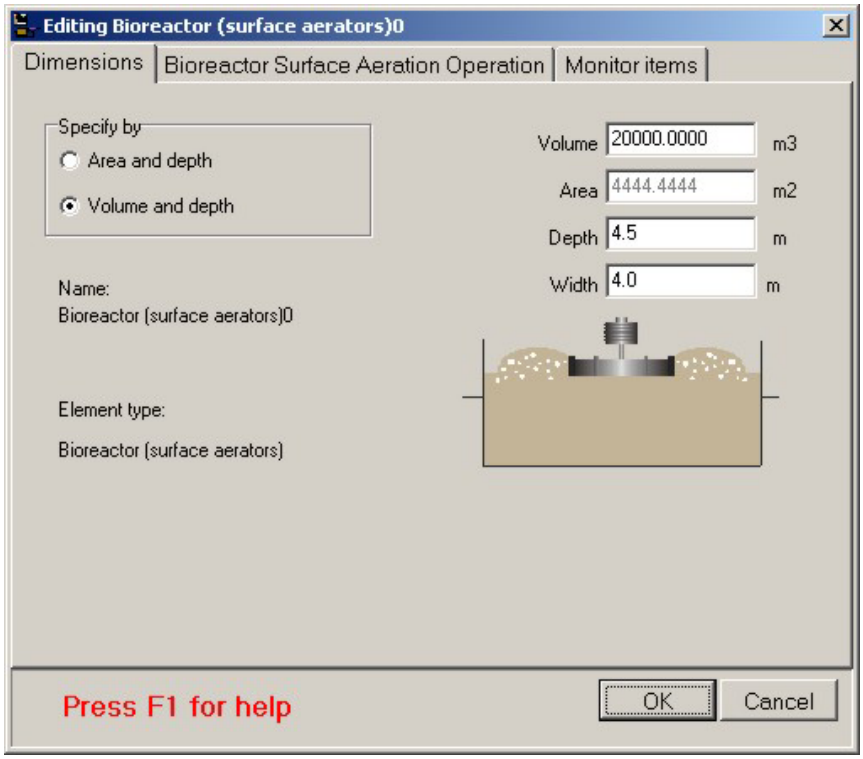
The surface aerator bioreactor simulates the activated sludge process in a continuous stirred tank reactor (CSTR) similarly to a regular bioreactor. However, aeration may be entered in terms of power supply rather than air supply. You may specify parameters related to the operation and control of the surface aerator bioreactor element. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The surface aerator bioreactor monitor items tab

Surface Aerator Bioreactor Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of a surface aerator bioreactor element.

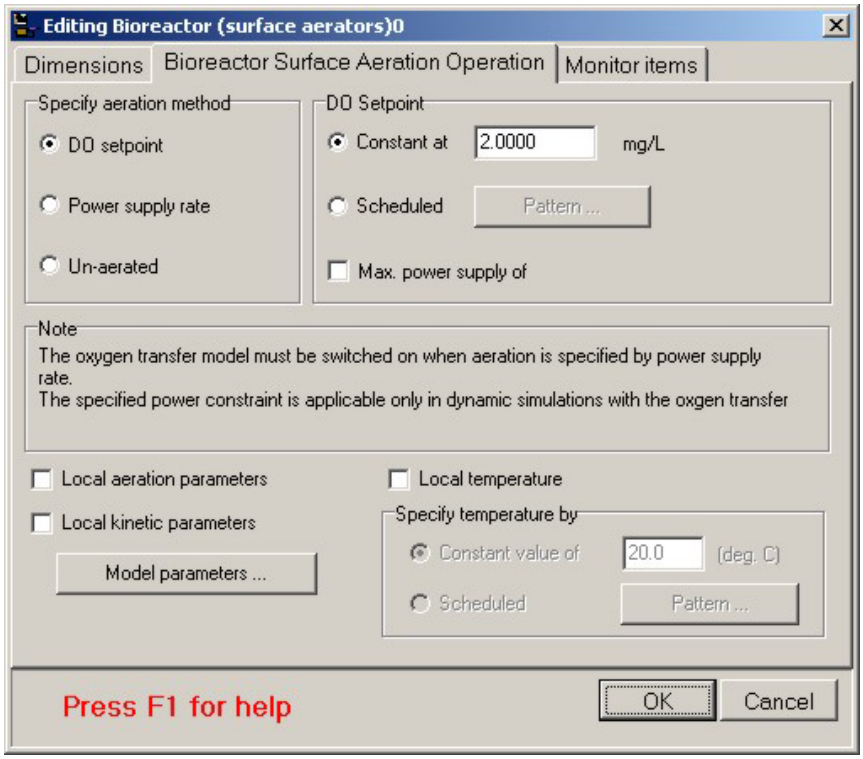


The surface aerator bioreactor dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the surface aerator bioreactor area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the surface aerator bioreactor volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a **Width** for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

Surface Aerator Bioreactor Operation

The **Operation** tab, shown below, allows the user to enter operating parameters for a surface aerator bioreactor element, such as aeration specifications and local temperature, as well as change local model parameters.



The surface aerator bioreactor operation tab

There are two methods for specifying aeration: either by a **DO setpoint** or by a **Power supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **Edit DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable power supply rate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Power supply rate** is selected, the power supply rate must be specified. You may specify either a **Constant** power supply rate or a **Scheduled** power supply rate pattern (clicking the **Pattern…** button will open the **Edit power supply rate itinerary** dialog box).

Note that if you wish to change any of the aeration model parameters, click the **Model parameters…** button to open the **Model parameter editor** dialog box (Note that this changes the aeration model parameters on a local level only).

A **Local temperature** also may be specified for a surface aerator bioreactor element. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

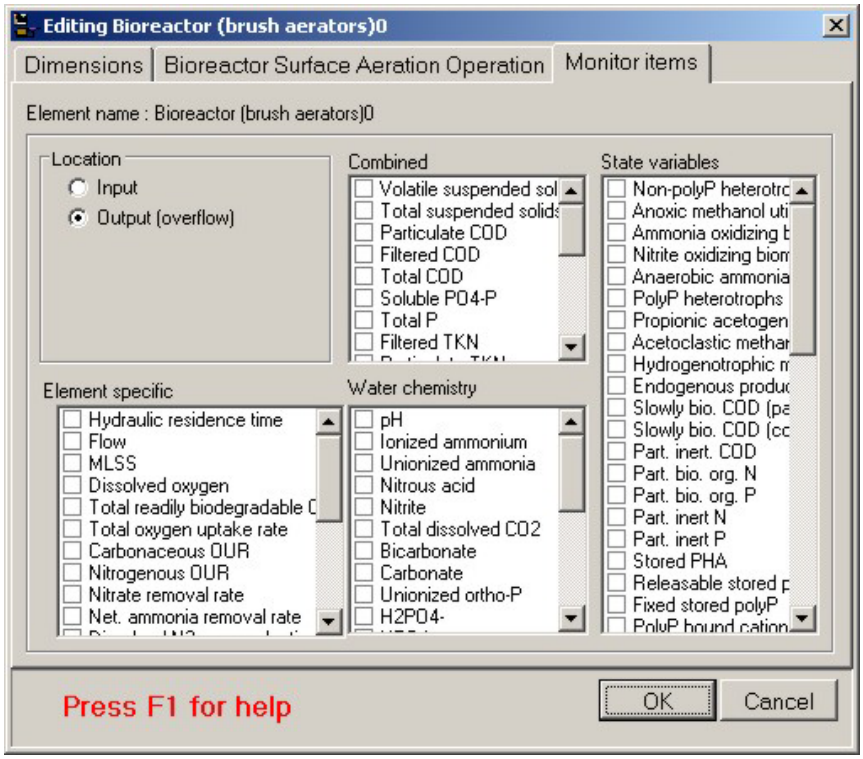
You also can specify **Local kinetic parameters** for the activated sludge model used in the surface aerator bioreactor element. If you click on the check box for local kinetic parameters, then clicking the **Model parameters…** button opens the

Model parameter editor dialog box allowing you access to the various activated sludge model kinetic parameters.

To specify local aeration parameters (e.g. alpha and beta factors), check the box labeled **Local aeration parameters**.

Brush Aerator Bioreactor

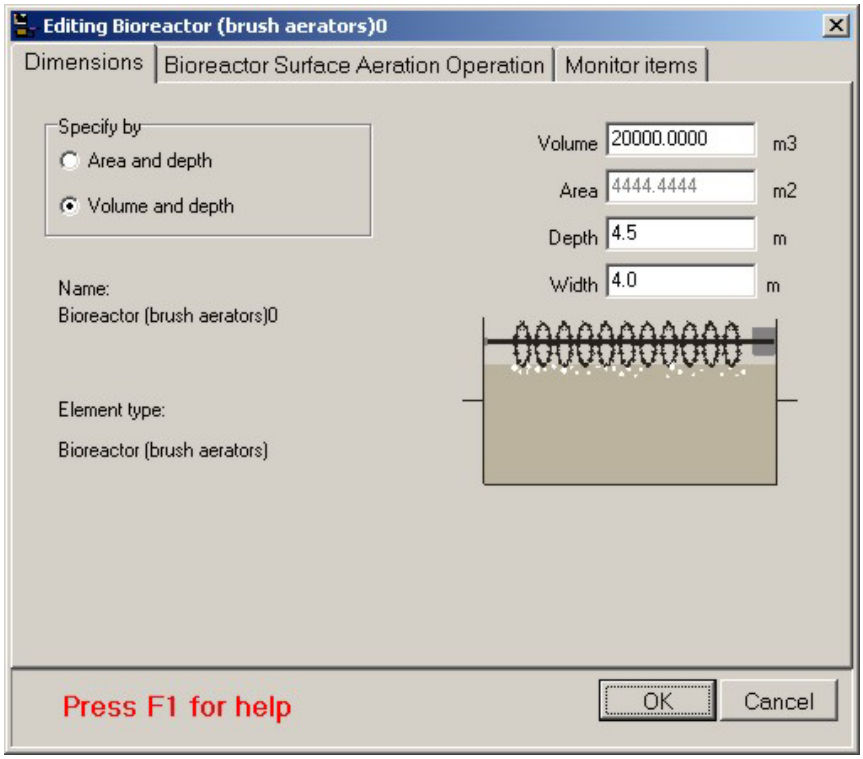
The brush aerator bioreactor simulates the activated sludge process in a continuous stirred tank reactor (CSTR) similarly to a regular bioreactor. However, aeration may be entered in terms of power supply rather than air supply. You may specify parameters related to the operation and control of the brush aerator bioreactor element. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The brush aerator bioreactor monitor items tab

Brush Aerator Bioreactor Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of a brush aerator bioreactor element.

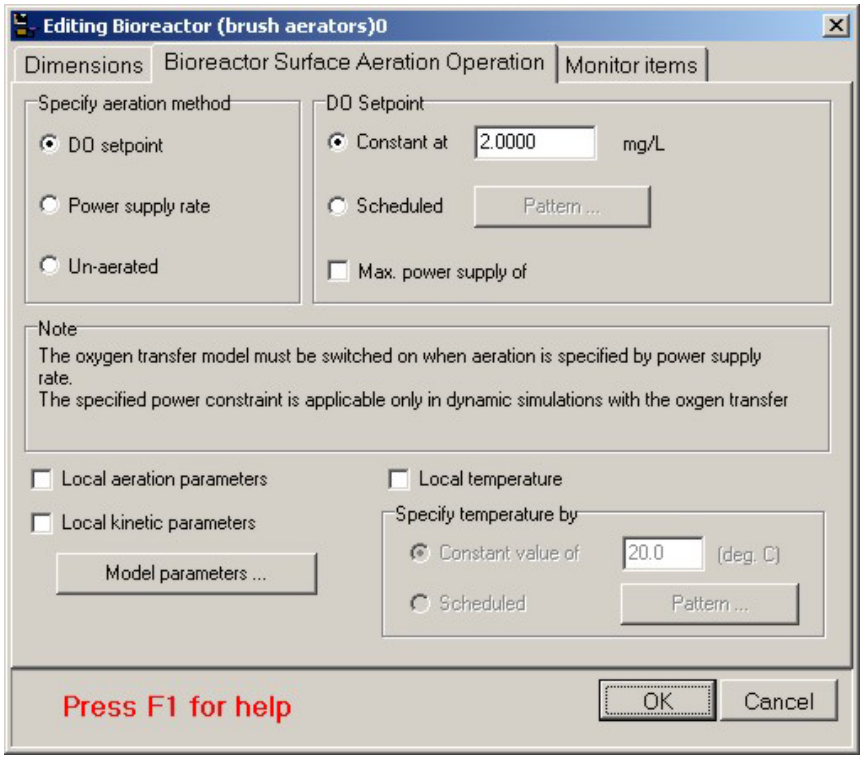


The surface aerator bioreactor dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the brush aerator bioreactor area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the brush aerator bioreactor volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a **Width** for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

Brush Aerator Bioreactor Operation

The **Operation** tab, shown below, allows the user to enter operating parameters for a brush aerator bioreactor element, such as aeration specifications and local temperature, as well as change local model parameters.



The brush aerator bioreactor operation tab

There are two methods for specifying aeration: either by a **DO setpoint** or by a **Power supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **Edit DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable power supply rate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Power supply rate** is selected, the power supply rate must be specified. You may specify either a **Constant** power supply rate or a **Scheduled** power supply rate pattern (clicking the **Pattern…** button will open the **Edit power supply rate itinerary** dialog box).

Note that if you wish to change any of the aeration model parameters, click the **Model parameters…** button to open the **Model parameter editor** dialog box (Note that this changes the aeration model parameters on a local level only).

A **Local temperature** also may be specified for a brush aerator bioreactor element. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

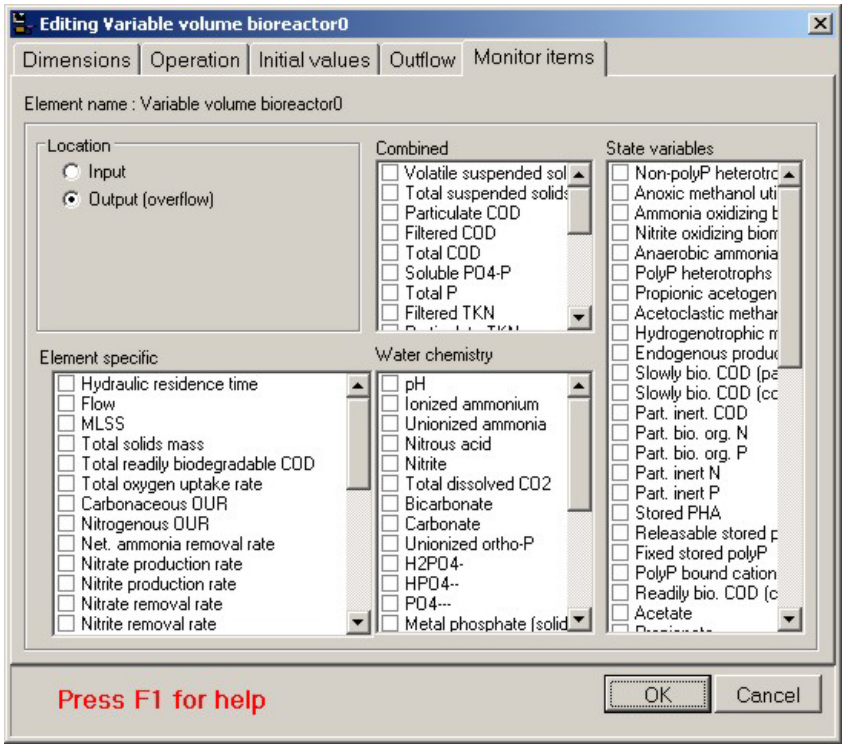
You also can specify **Local kinetic parameters** for the activated sludge model used in the brush aerator bioreactor element. If you click on the check box for local kinetic parameters, then clicking the **Model parameters…** button opens the

Model parameter editor dialog box allowing you access to the various activated sludge model kinetic parameters.

To specify local aeration parameters (e.g. alpha and beta factors), check the box labeled **Local aeration parameters**.

Variable Volume/Batch Bioreactor

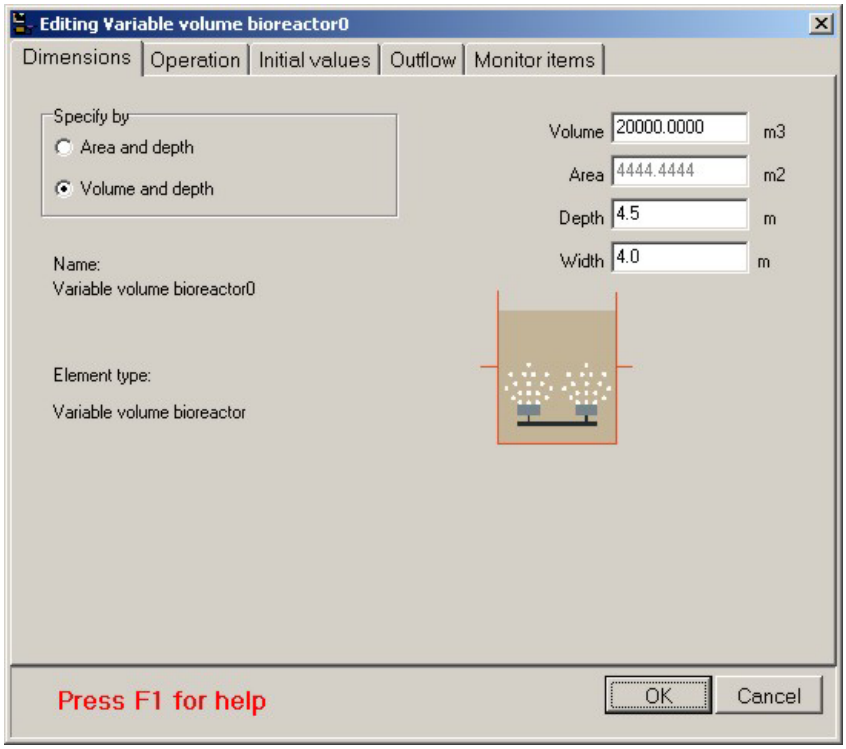
The variable volume/batch bioreactor simulates the activated sludge process in a variable volume continuous stirred tank reactor (CSTR). You can specify parameters related to the operation and control of the variable volume/batch bioreactor element, as well as parameters related to the initial conditions. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The variable volume/batch bioreactor monitor items tab

Variable Volume/Batch Bioreactor Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of a variable volume/batch bioreactor element.

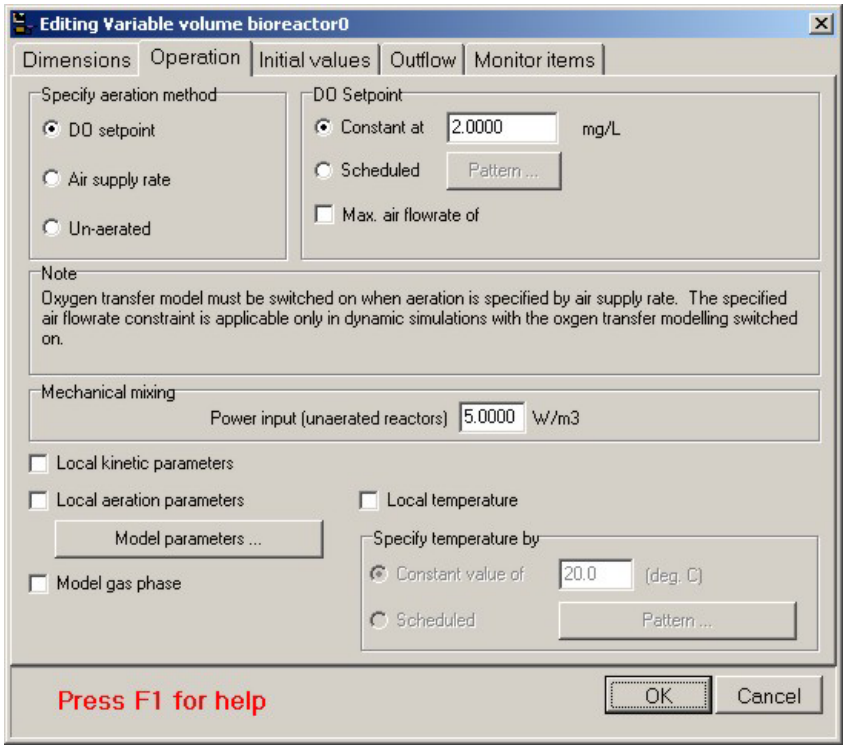


The variable volume/batch bioreactor dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the variable volume/batch bioreactor area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the variable volume/batch bioreactor volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a **Width** for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

Variable Volume/Batch Bioreactor Operation

The **Operation** tab, shown below, allows the user to enter operating parameters for a variable volume/batch bioreactor element, such as aeration specifications and local temperature, as well as change local model parameters.



The variable volume/batch bioreactor operation tab

There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **Edit DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **Edit air flow itinerary** dialog box.

The mechanical mixing **Power input (unaerated reactors)** can be specified. This is used to calculate the velocity gradient in the vessel.

Note : If you wish to change any of the aeration model parameters, click the **Model parameters…** button to open the **Model parameter editor** dialog box (Note that this changes the aeration model parameters on a local level only).

A **Local temperature** also may be specified for a variable volume/batch bioreactor element. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

You also can specify **Local kinetic parameters** for the activated sludge model used in the variable volume/batch bioreactor element. If you click on the check box

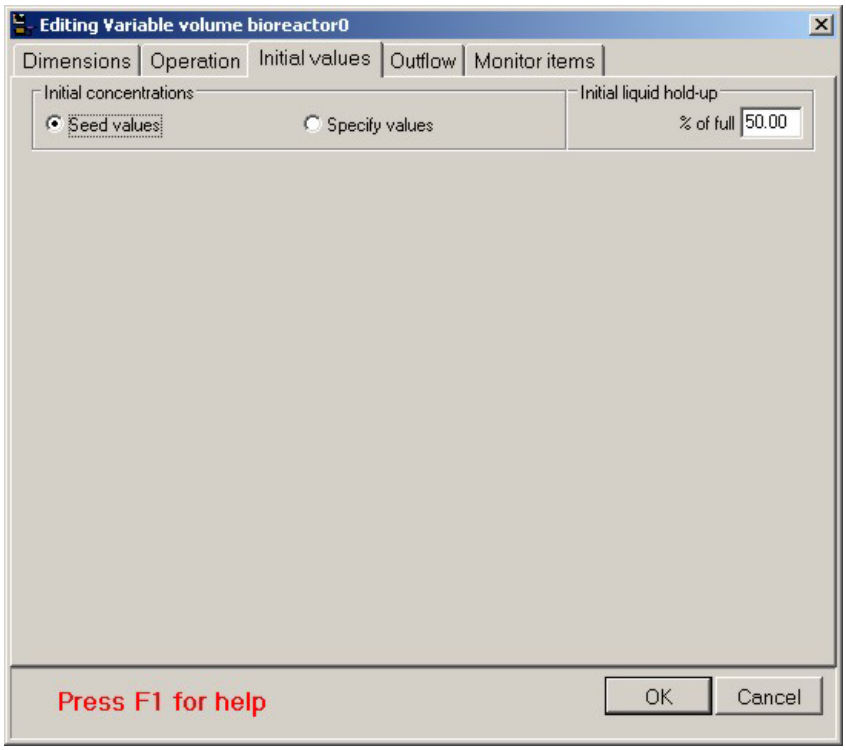
for local kinetic parameters, then clicking the **Model parameters…** button opens the **Model parameter editor** dialog box allowing you access to the various activated sludge model kinetic parameters.

To specify local aeration parameters (e.g. alpha and beta factors), check the box labeled **Local aeration parameters**.

To model the concentrations of the constituents of the aeration gas (e.g. oxygen, carbon dioxide, etc.), check the box labeled **Model gas phase**. If this option is selected you may also enter a percentage value for the **Gas hold-up** (i.e. the percentage of the total reactor volume occupied by aeration gas).

Variable Volume/Batch Bioreactor Initial Values

The **Initial values** tab, shown below, is used for setting up the initial settings for variable volume/batch bioreactor concentrations and volume.



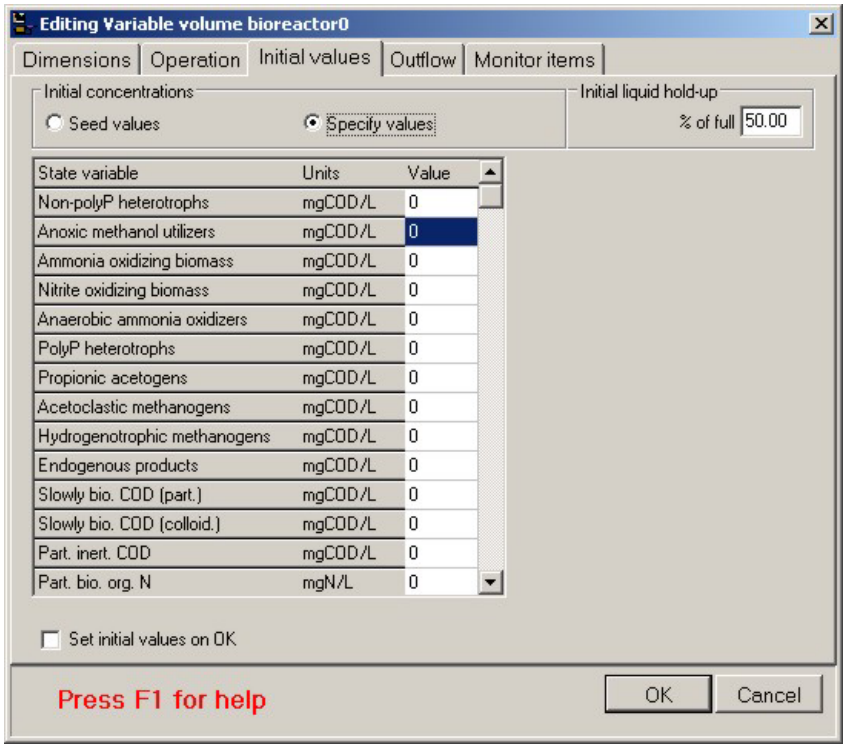
The variable volume/batch bioreactor initial values tab with the Seed values *option active*

Use the **Initial liquid hold-up** field to enter a value to specify the **% of full** setting. This initial liquid hold-up volume will be used for steady state calculations. It should be noted here that the lower and upper limits on this value are 0.02 and 99.98%, respectively. This reflects the fact that this is the liquid volume, and allows for small differences between *liquid* volume and *reactor* volume for inlet and outlet pipes.

There are two methods for setting up **Initial concentrations** in the variable volume/batch bioreactor. The dialog box shown above illustrates the first case where the **Seed values** option is selected. When this option is selected, BioWin applies default seed values for all the state variables (except volume, which you specify) in the same manner it would for other bioreactor elements.

The dialog box shown below illustrates the second case where the **Specify values** option is selected. In this case, an editable list of state variables is displayed in the

dialog box. This allows you to enter your own seed values for state variables in the variable volume/batch bioreactor. These will be inserted in the variable volume/batch bioreactor when you begin a steady state simulation, or when you begin a dynamic simulation and choose to use seed values.



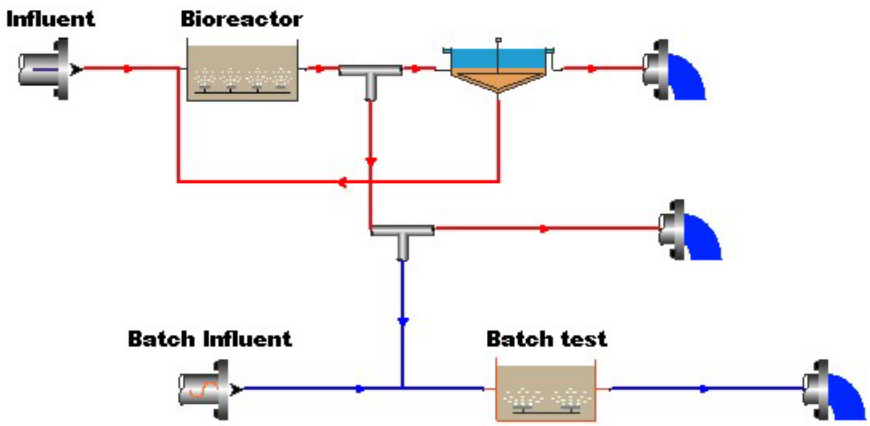
The variable volume/batch bioreactor initial values tab with the Specify values *option active*

When the specified seed values get inserted in the variable volume/batch bioreactor is determined by the **Set initial values on OK** option. If this option **is not** checked:

- The specified seed values will be placed in the variable volume/batch bioreactor at the beginning of a steady state simulation if you elect to start from seed conditions;
- The specified values will be placed in the variable volume/batch bioreactor at the beginning of a dynamic simulation **if** you elect to seed the simulation.

If this option **is** checked, the specified values are placed in the variable volume/batch bioreactor when you click **OK** to close the dialog box, **overriding any existing state variable values**. Therefore, the state variable values at the beginning of a dynamic simulation will be the same regardless of whether or not you choose to re-seed.

The use of this option is best explained by the example illustrated below. In this example, we want to feed a variable volume/batch bioreactor (the bottom train with blue pipes) with a portion of the waste from a normal bioreactor that is running at steady state conditions (i.e. the top train with red pipes). However, we don't want the variable volume/batch bioreactor to start up at steady state – we want to specify start-up conditions and then investigate the dynamic behavior.



Example system illustrating the use of variable volume / batch bioreactor seeding options

To obtain steady state conditions, set up the configuration and run a steady state simulation. Next, on the variable volume/batch bioreactor **Initial values** tab, select the **Specify values** option for initial concentrations and enter the desired state variable start-up conditions for the batch test. Select the **Set initial values on OK** option so that the start-up values are immediately placed in the variable volume/batch bioreactor. Now when the dynamic simulation is started, choose to use the **Current values** in elements to begin. In this example, the **Set initial values on OK** option is what allows us to have unique start-up conditions in the batch test.

Variable Volume/Batch Bioreactor Outflow

Remember for steady state simulations, the initial liquid hold-up volume will be used for calculations.

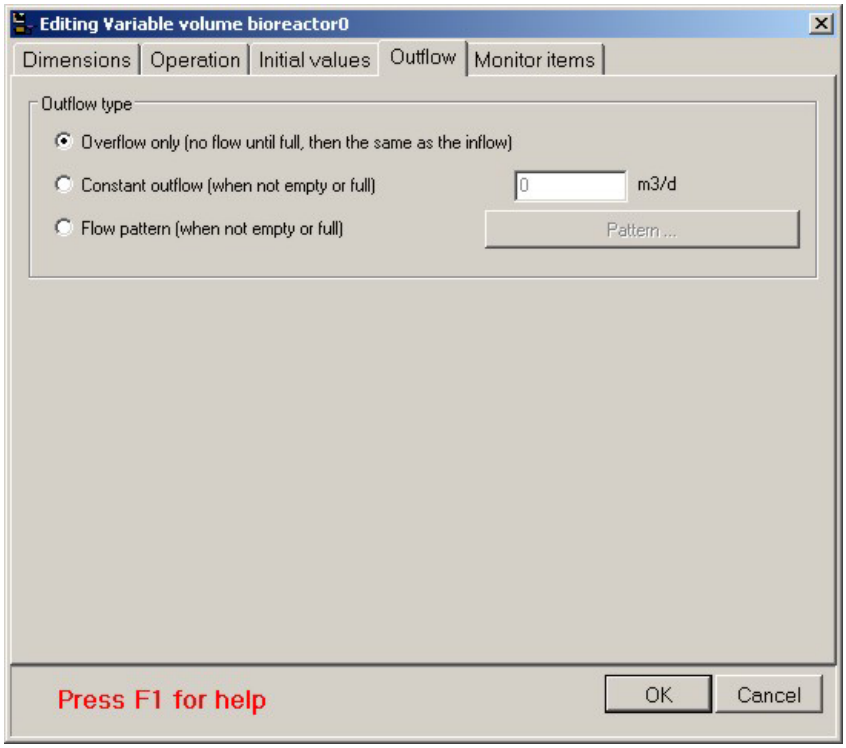
The variable volume/batch bioreactor element **Outflow** tab, shown below, is used to specify the overflow behavior. The overflow behavior can be generalized as follows:

- Whenever the variable volume/batch bioreactor is full, it overflows at the influent rate, regardless of the overflow setting.
- Whenever the variable volume/batch bioreactor is empty and the outflow rate is set higher than the influent rate, the variable volume/batch bioreactor will only have an outflow equal to the influent flow, so as not to have negative volume. If the outflow rate is set lower than the influent rate, then the variable volume/batch bioreactor will begin to fill up.

The following is a description of the behavior of the three outflow settings:

- **Overflow only** – the variable volume/batch bioreactor fills up, and then overflows at the influent flow rate (this setting would be used to simulate start-up of a bioreactor).
- **Constant outflow** – the outflow always tries to attain the specified constant rate, except when physically constrained (i.e. when the variable volume/batch bioreactor is full or empty)
- **Flow pattern** – the outflow always tries to attain the current specified pattern rate, except when physically constrained (i.e. when the variable volume/batch bioreactor is full or empty). To specify a pattern, click

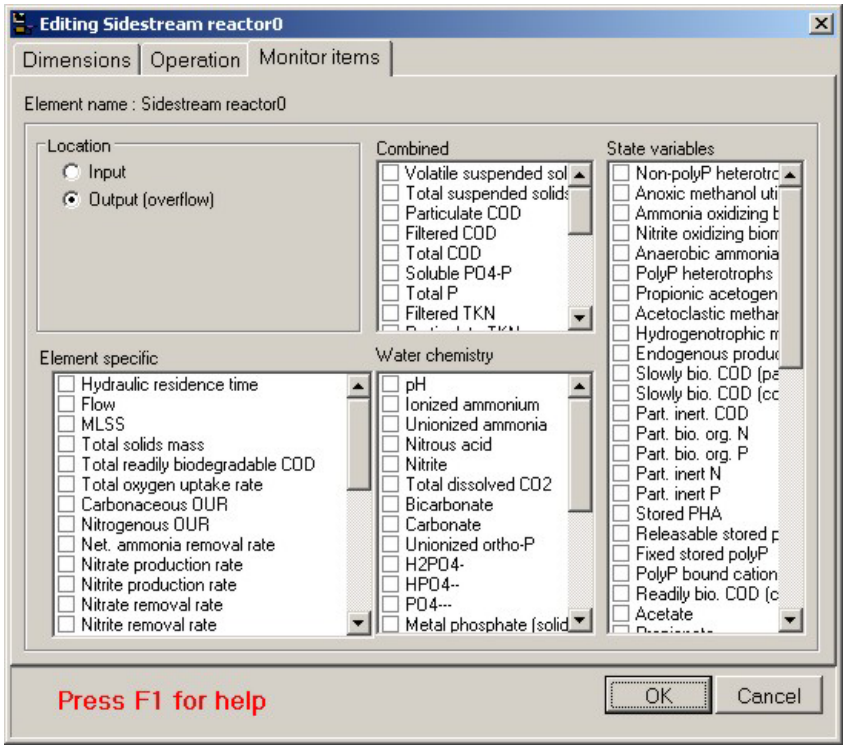
the **Pattern…** button when it becomes active. For more information , see the **Liquid outflow itinerary** section.



The variable volume/batch bioreactor outflow settings tab

Sidestream reactor

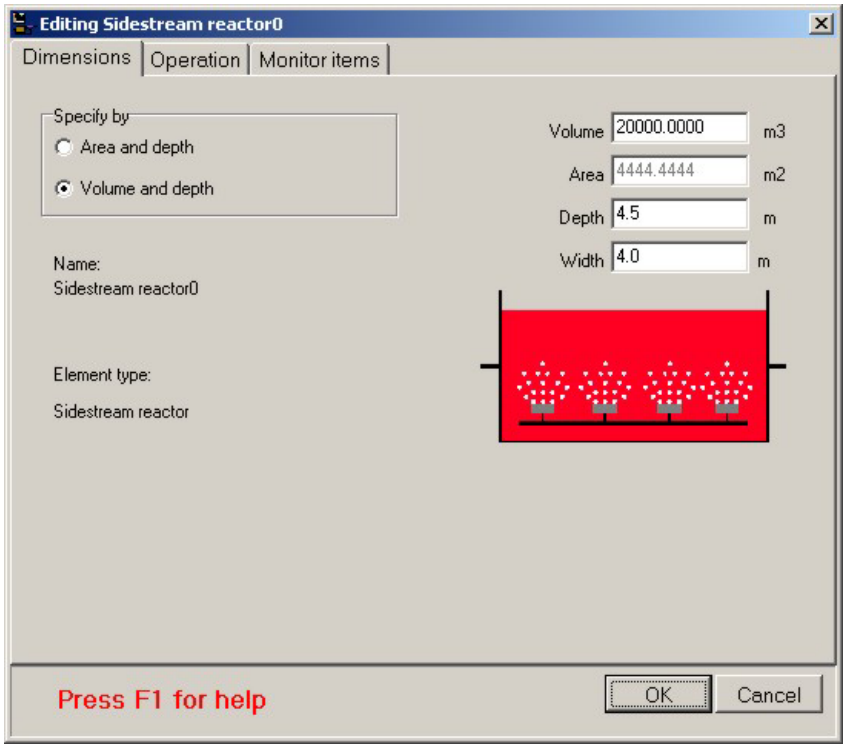
The sidestream reactor simulates the activated sludge process in a continuous stirred tank reactor (CSTR). Complex activated sludge systems can be configured by arranging a number of sidestream reactors in series/parallel. You may specify parameters related to the operation and control of the sidestream reactor element. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The monitor items tab

Sidestream Reactor Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of a sidestream reactor element.

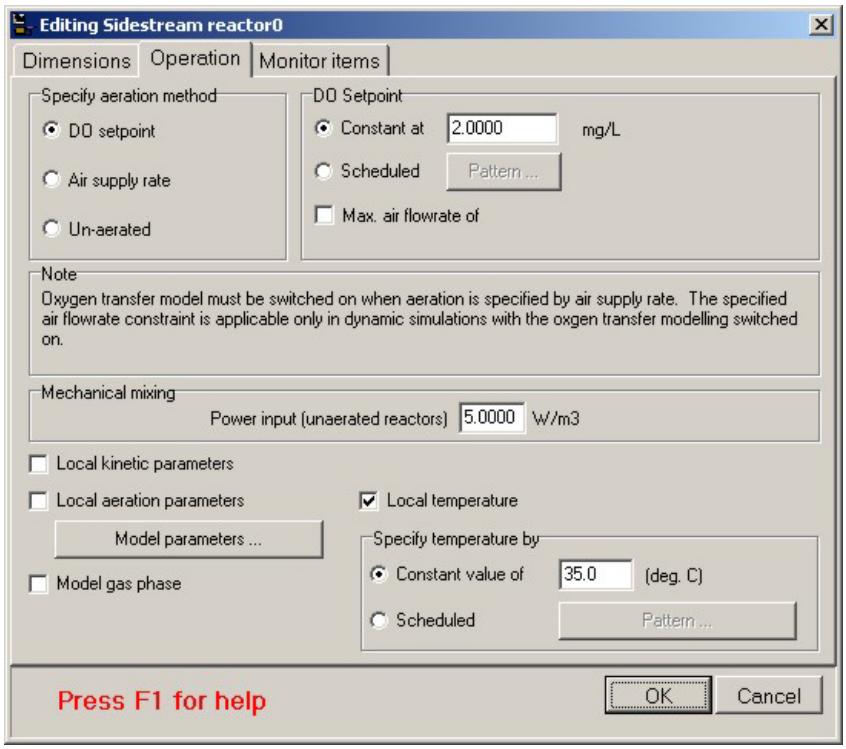


The dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the sidestream reactor area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the sidestream reactor volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a **Width** for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

Sidestream Reactor Operation

The **Operation** tab, shown below, allows the user to enter operating parameters for a sidestream reactor element, such as aeration specifications and local temperature, as well as change local model parameters.



The operation tab

There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **Edit DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **Edit air flow itinerary** dialog box).

The mechanical mixing **Power input (unaerated reactors)** can be specified. This is used to calculate the velocity gradient in the vessel.

Note : If you wish to change any of the aeration model parameters, click the **Model parameters…** button to open the **Model parameter editor** dialog box (Note that this changes the aeration model parameters on a local level only).

A sidestream reactor element has the **Local temperature** option selected by default as if is frequently used to model digester effluents that may be elevated in temperature. Although the default setting is a constant temperature of 35 degrees celcius the **Specify temperature by** radio button allows you to specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

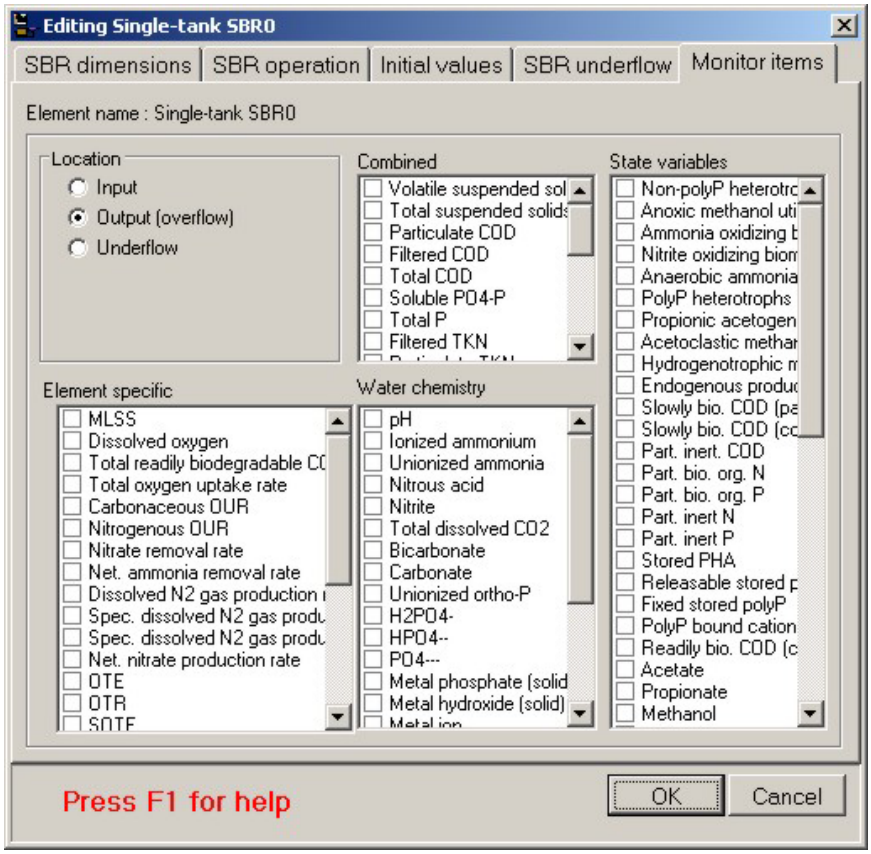
You also can specify **Local kinetic parameters** for the activated sludge model used in the sidestream reactor element. If you click on the check box for local kinetic parameters, then clicking the **Model parameters…** button opens the **Model parameter editor** dialog box allowing you access to the various activated sludge model kinetic parameters.

To specify local aeration parameters (e.g. alpha and beta factors), check the box labeled **Local aeration parameters**.

To model the concentrations of the constituents of the aeration gas (e.g. oxygen, carbon dioxide, etc.), check the box labeled **Model gas phase**. If this option is selected you may also enter a percentage value for the **Gas hold-up** (i.e. the percentage of the total reactor volume occupied by aeration gas).

Single-Tank Sequencing Batch Reactor

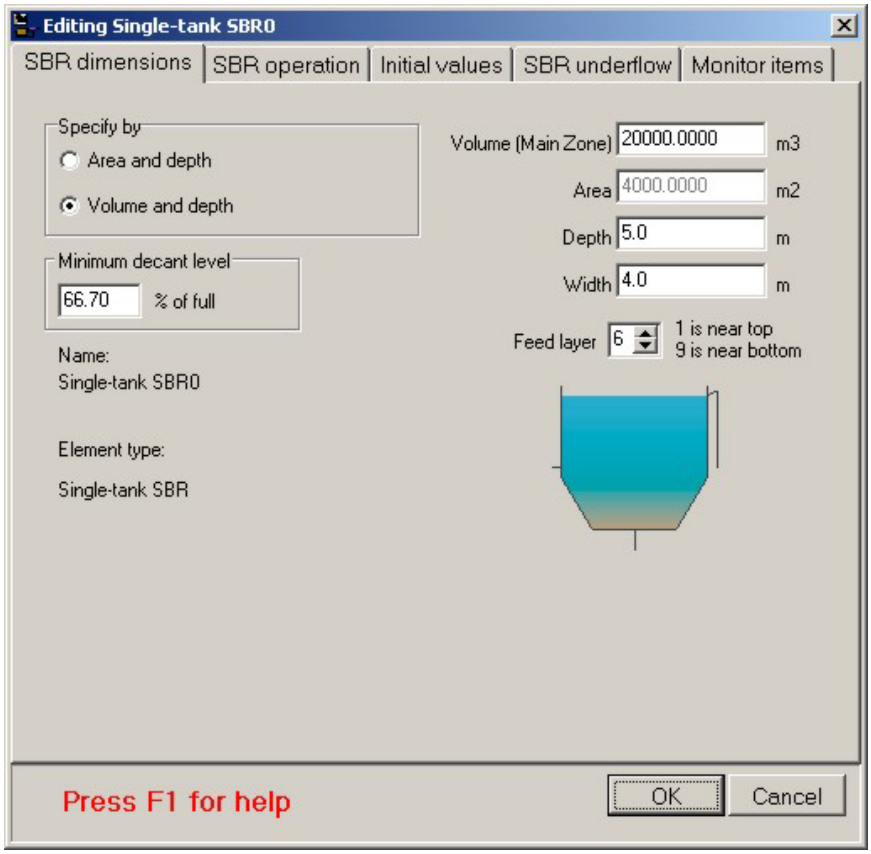
The single-tank sequencing batch reactor (STSBR) is the simplest SBR element available in BioWin. However, it still can be a complex unit to incorporate into a configuration. You may specify parameters related to the STSBR dimensions, cycle operation, starting values, and underflow rates. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The STSBR monitor items tab

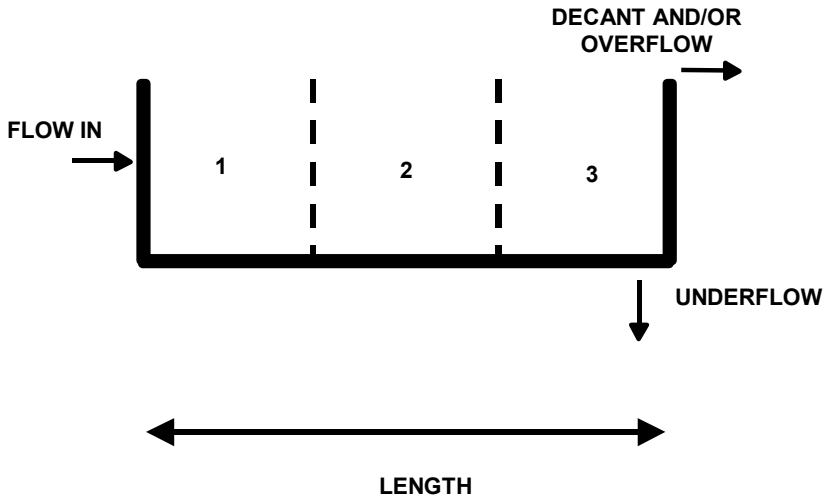
STSBR Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of a STSBR element.



The STSBR dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the STSBR area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the STSBR volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Units are shown to the right of the edit boxes. Regardless of the method you choose, you also must specify a **Width** for the STSBR.



Flow distribution in a Single-tank SBR

BioWin uses the **Width** parameter to calculate the length of the STSBR. The length is then divided into three equally sized zones. Flow enters the first zone at the level specified by the **Feed layer**. Underflow leaves the STSBR at the bottom of the third zone, and decant and/or overflow leaves the STSBR at the top of the third zone.

Some further explanation of how BioWin uses the SBR dimensions is warranted. When you set the dimensions, you specify a **Volume**, **Depth** and **Width** (and **Length** is calculated). Now think of a 2-d side-on view of the SBR with flow from left to right, as shown above.

For example, imagine the full depth is 5 m and the length is 15 m. When settling starts, this is what happens:

- The horizontal length of the SBR is divided into three equal-length subsections (each 5 m in this case).
- The vertical direction is divided into 10 equal-depth layers. [**Note:** as the level goes down during decanting - with no feed - the number of layers stays at 10, and the depth of each layer decreases]. That is, think of each length section as being 10 layers stacked on top of each other (and there are 3 sections).
- Each of the three sections is treated as a vertical settler. That is, the SBR (during settling) is 3 side-by-side settlers, with a total of 30 cells (each completely mixed i.e. of uniform composition).

Consider the dimensions of each of the 30 cells:

- The horizontal area of each cell is **Width * (Length/3) = W * 5 m** here.
- The end area of each cell is **Width * (Liquid Depth/10) = W * 0.5 m** (0.5 m when it is full). [**Note:** in this case the ratio of cell horizontal to

side area is 5 : 0.5 = 10 : 1 (if tank is full - but ratio increases when tank level drops)].

For the moment. assume that there is no inflow during decanting. Decant is removed from the top-right cell. During decanting several things happen (apart from settling in each of the 3 sections):

- Say 1 (incremental) unit of volume is removed from the top-right cell (at the composition of the top right cell).
- Overall, the volume of each of the 30 cells must be decreased by 1 unit / 30 (the length of each cell is fixed, so the depth must decrease).
- 29/30ths of the unit volume must get into the top right cell. That is, overall 1/30th of the volume must be removed from each of the other 29 cells.
- The 29/30ths gets into the top right cell from the cell to the left and from the cell below it.
- The cell at the bottom left of the SBR is a special case - there is no flow into it - only outflow of 1/30th. This flow leaves via the top surface and via the right hand end surface. The flow leaving each of the 2 surfaces is proportional to the areas of each surface.
- Flow balances around each of the other 28 cells is handled in a similar manner, doing a simultaneous flow balance on each cell. For the typical cell, there can be:
    - Flow out of the top surface;
    - Flow out the right side;
    - Flow in through the bottom surface; and
    - Flow in through the left surface.

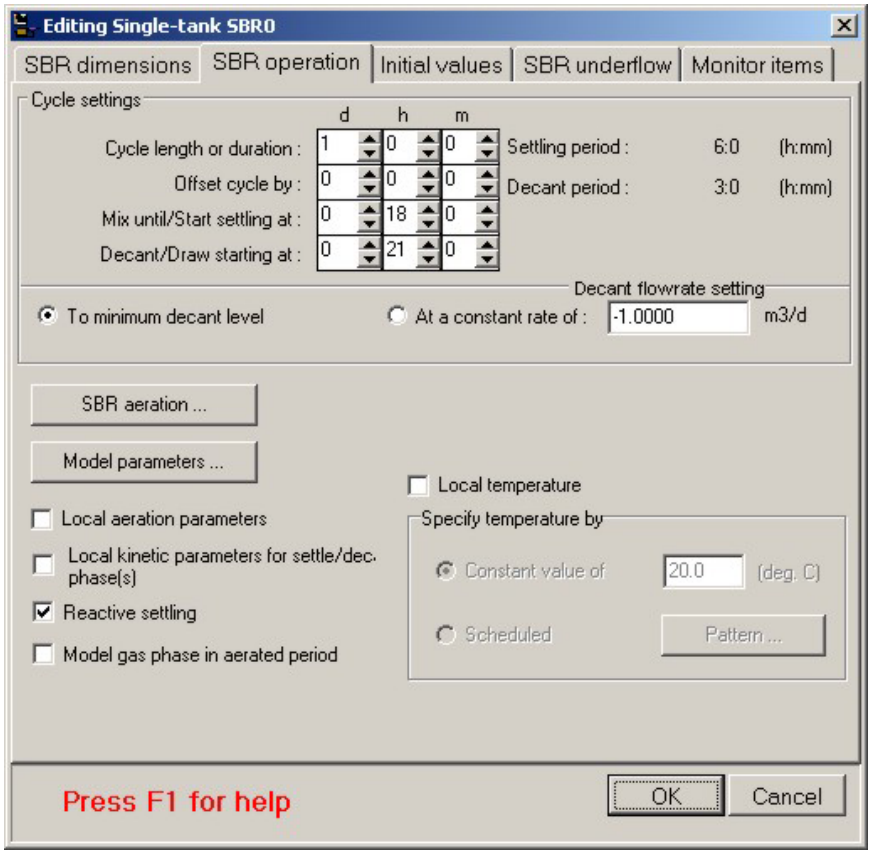
If there is influent flow during decanting, the flow balance is just slightly more complex, but happens in a similar fashion. Note that this explanation makes the process sound like a step-by-step calculation. In reality, what happens is a whole series of simultaneous integrations with error checking.

A **Minimum decant level** must also be specified as a percentage of the total STSBR volume. This level is the lowest liquid level that can be achieved by decanting liquid out of the top of the STSBR. Note that it *is* possible to obtain lower liquid levels in a cycle than this value, but to get lower than this level, the liquid would have to go out via the underflow.

The **Feed layer** also is specified on this tab. This is the layer where liquid flow is introduced to the element. Note that the layers are numbered such that layer 1 is near the top, and number 9 is near the bottom. The feed flow is distributed throughout the STSBR according to the vessel geometry.

STSBR Operation

The **Operation** tab, shown below, allows the user to enter operating parameters for a STSBR element, such as cycle patterns, decant flow rates, aeration specifications and local temperature, as well as change local model parameters.



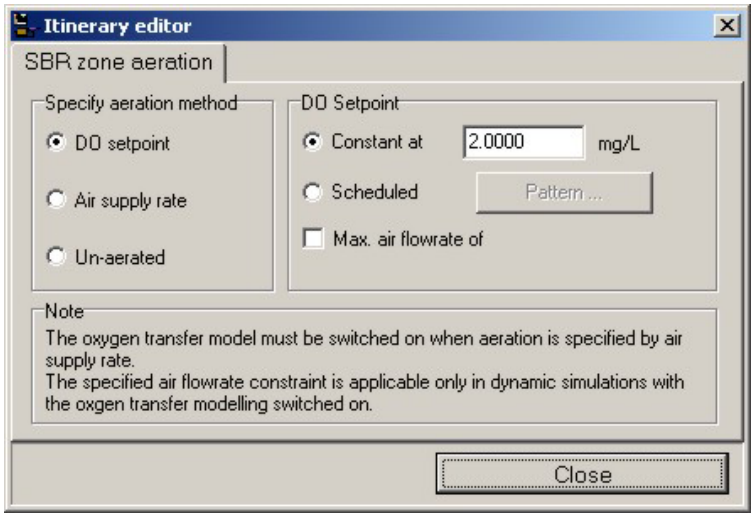
The STSBR operation tab

The major operational settings of the STSBR are those related to **Cycle settings**. The **Cycle length or duration** is the total time that will elapse before the cycle begins to repeat itself. The first event to take place in the cycle is the end of the mixing phase/beginning of the settling phase. You specify this event time with the **Mix until/Start settling at:** spin edit control. Note that the dialog provides you with feedback as to the length of your settling period The second event to take place is the beginning of the decant phase. You specify this event time with the **Decant/Draw starting at:** spin edit control. Note that the dialog provides you with feedback as to the length of your decant period. Experimenting with these settings will reveal that settling and decanting take place simultaneously – but the settling phase length always must be greater than the decant phase length. Finally, you may set up a cycle offset if you wish using the **Offset cycle by:** control.

Note that BioWin assumes that there is no influent or underflow activity during the decant period and bases the calculation for required decant flowrate on that premise.

The next operational setting is the **Decant flowrate setting**. You may specify this with one of two possible methods. If you know the flowrate you wish to decant at, then choose the **At a constant rate of:** option and enter the desired value. If you wish BioWin to calculate the decant flowrate for you, select the **To minimum decant level** option. The decant flowrate is calculated as follows. At the beginning of the decant period, the simulator looks at the difference between the current liquid volume and the specified minimum decant volume. It then calculates the necessary flowrate to decant the available liquid during the decant period.

You can specify the STSBR aeration settings by clicking the **SBR aeration…** button which will show the following dialog box.



The STSBR aeration setting dialog box

There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **SBR DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **SBR air flowrate itinerary** dialog box).

Note that if you wish to change any of the aeration model parameters, click the **Model parameters…** button in the main STSBR **Operation** tab to open the **Model parameter editor** dialog box (Note that this changes the aeration model parameters on a local level only)..

A **Local temperature** also may be specified for a STSBR element. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

You also can specify **Local kinetic parameters** for the activated sludge model used in the STSBR **decant/settling phase**. For the mixing phase, the global model parameter values are used. If you click on the check box for local kinetic parameters, then clicking the **Model parameters…** button opens the **Model parameter editor** dialog box allowing you access to the various activated sludge model kinetic parameters.

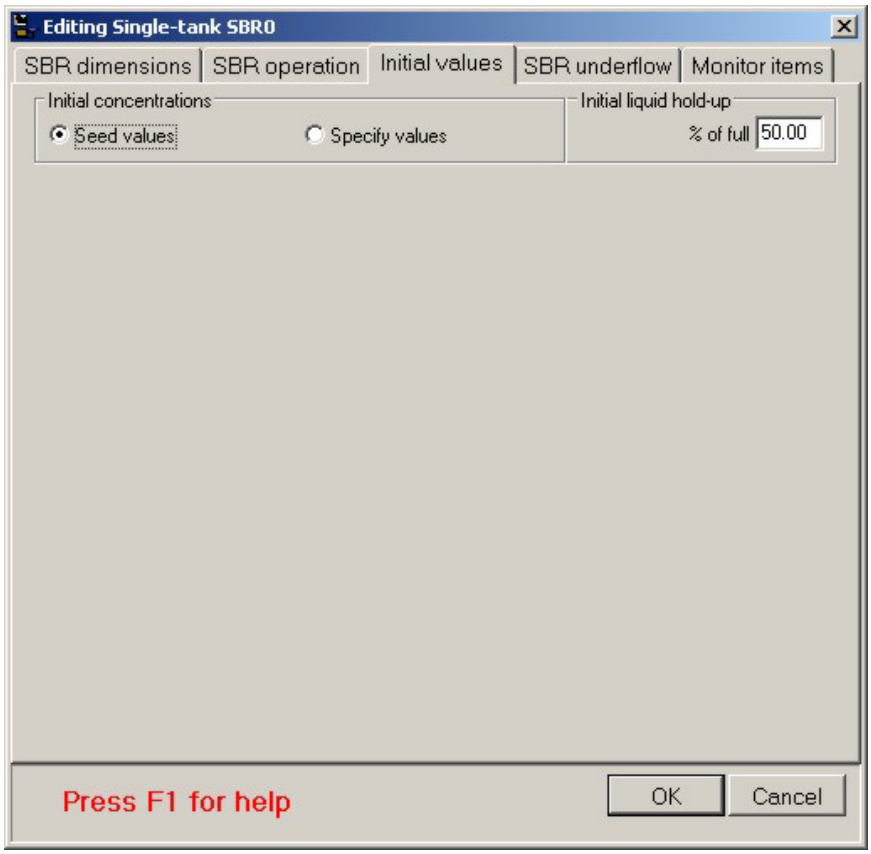
To specify local aeration parameters (e.g. alpha and beta factors), check the box labeled **Local aeration parameters**.

To model the concentrations of the constituents of the aeration gas (e.g. oxygen, carbon dioxide, etc.), check the box labeled **Model gas phase**. If this option is

selected you may also enter a percentage value for the **Gas hold-up** (i.e. the percentage of the total reactor volume occupied by aeration gas).

STSBR Initial Values

The **Initial values** tab, shown below, is used for setting up the initial settings for STSBR element concentrations and volume.

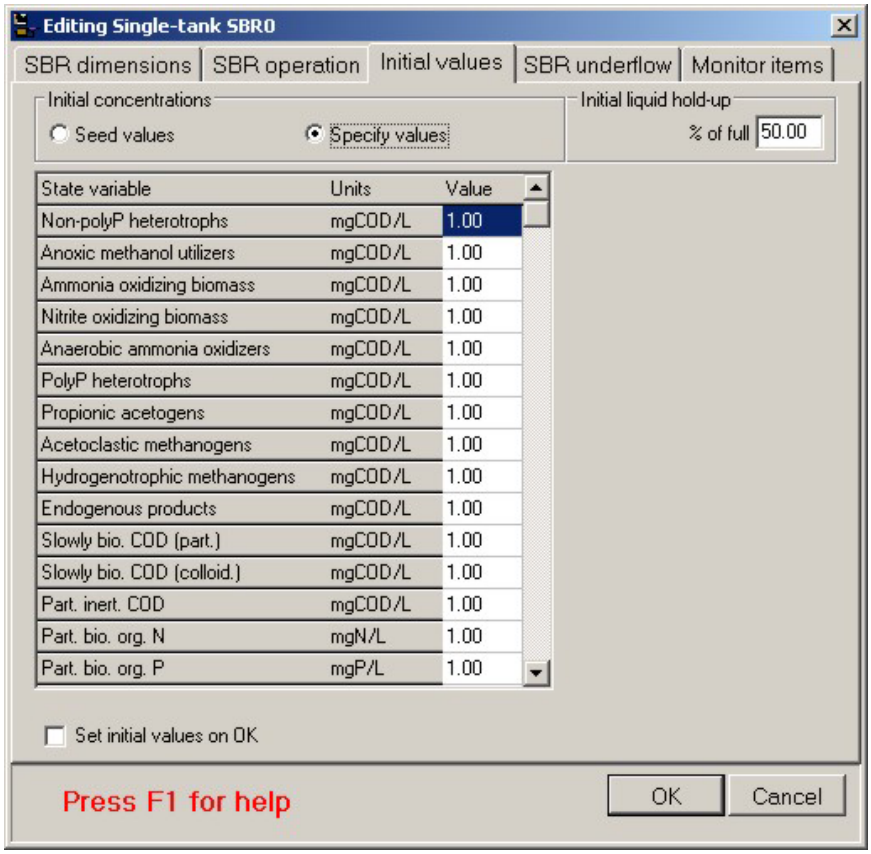


The STSBR initial values tab with the Seed values *option active*

Use the **Initial liquid hold-up** field to enter a value to specify the **% of full** setting. It should be noted here that the lower and upper limits on this value are 0.02 and 99.98%, respectively. This reflects the fact that this is the liquid volume, and allows for small differences between *liquid* volume and *reactor* volume for inlet and outlet pipes.

There are two methods for setting up **Initial concentrations** in the STSBR. The dialog box shown above illustrates the first case where the **Seed values** option is selected. When this option is selected, BioWin applies default seed values for all the state variables (except volume, which you specify) in the same manner it would for other bioreactor elements.

The dialog box shown below illustrates the second case where the **Specify values** option is selected. In this case, an editable list of state variables is displayed in the dialog box. This allows you to enter your own seed values for state variables in the STSBR. These will be inserted in the STSBR when you begin a dynamic simulation and choose to use seed values.



The STSBR initial values tab with the Specify values *option active*

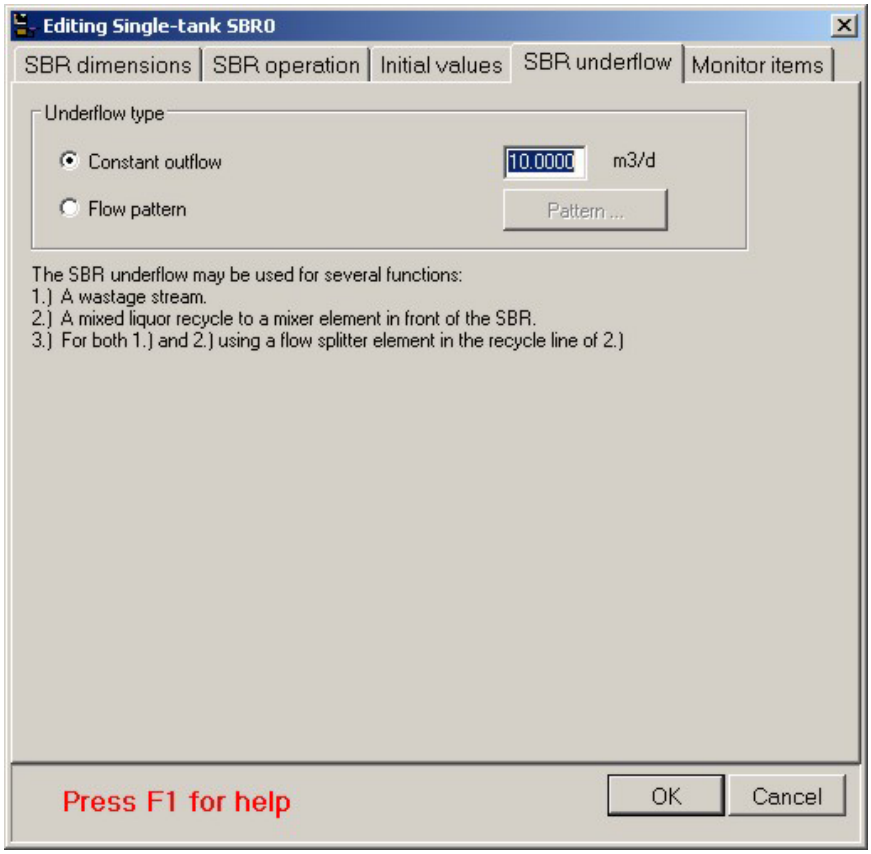
When the specified seed values get inserted in the STSBR is determined by the **Set initial values on OK** option. If this option **is not** checked:

- The specified values will be placed in the STSBR at the beginning of a dynamic simulation if you elect to seed the simulation.

If this option **is** checked, the specified values are placed in the STSBR when you click **OK** to close the dialog box, **overriding any existing state variable values**. Therefore, the state variable values at the beginning of a dynamic simulation will be the same regardless of whether or not you choose to re-seed.

STSBR Underflow

The STSBR **Underflow** tab shown below allows you to set up underflow rates and patterns for the STSBR element.



The STSBR underflow settings tab

The **Underflow type** may be specified as a **Constant underflow** or a **Flow pattern** by selecting the appropriate option. If you select **Constant underflow**, you may enter the desired underflow rate in the edit field. If you select **Flow pattern**, clicking the **Pattern…** button will open the **Underflow rate itinerary** dialog box.

The STSBR underflow may be used for several functions:

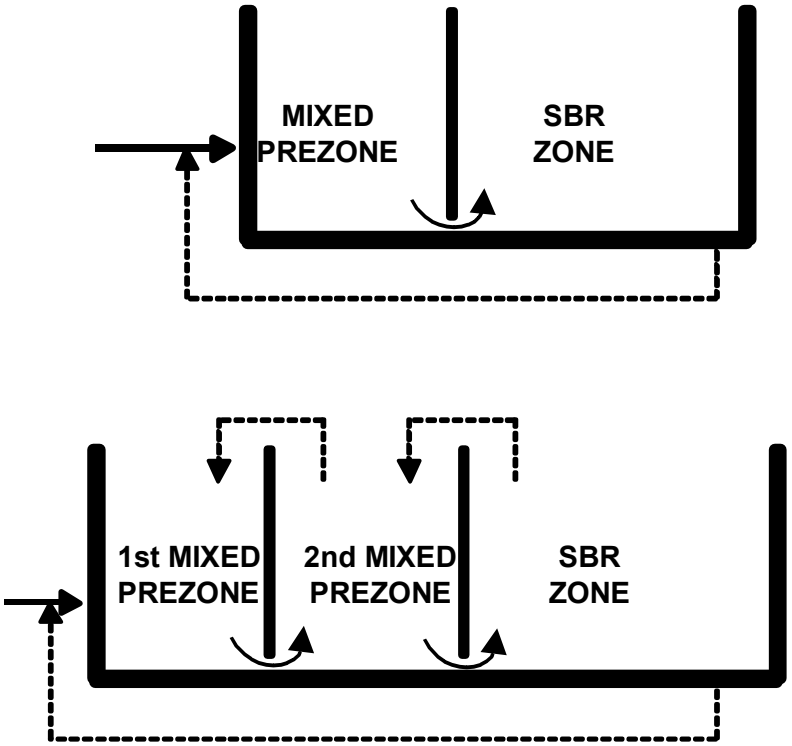
1. As a wastage stream;
2. A mixed liquor recycle to a mixer element in front of the STSBR;
3. As both a wastage and a recycle by placing a splitter element between the underflow and the mixer element in front of the STSBR.

SBR with Always-Mixed Prezone(s)

This type of SBR allows you to have hydraulically-linked prezones that are continuously mixed, even when the decant zone is in settle mode. The following SBR/always-mixed prezone combinations are available:

- SBR+1 Prezone
- SBR+2 Prezones

BioWin allows you to have internal recycles between the prezones and the SBR, as shown in the following diagram.

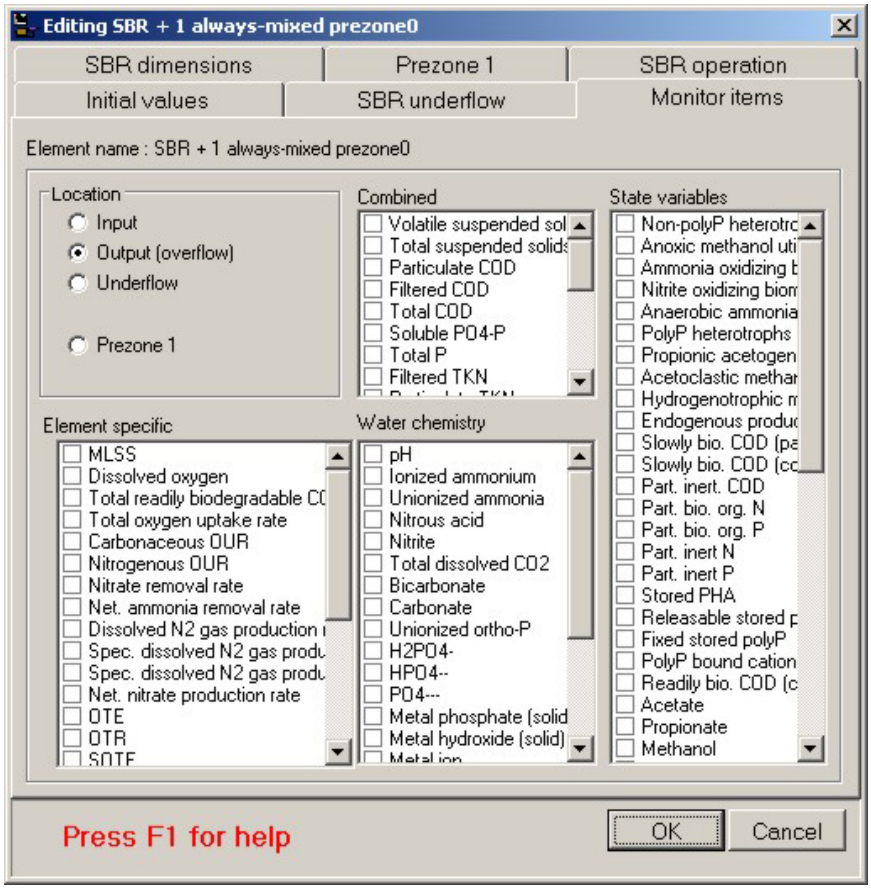


Flow recycles and patterns that can be set up between an SBR and its prezone(s)

The following sections describe the dialog boxes for the SBR with one and two always-mixed prezone(s), respectively.

SBR with One Always-Mixed Prezone

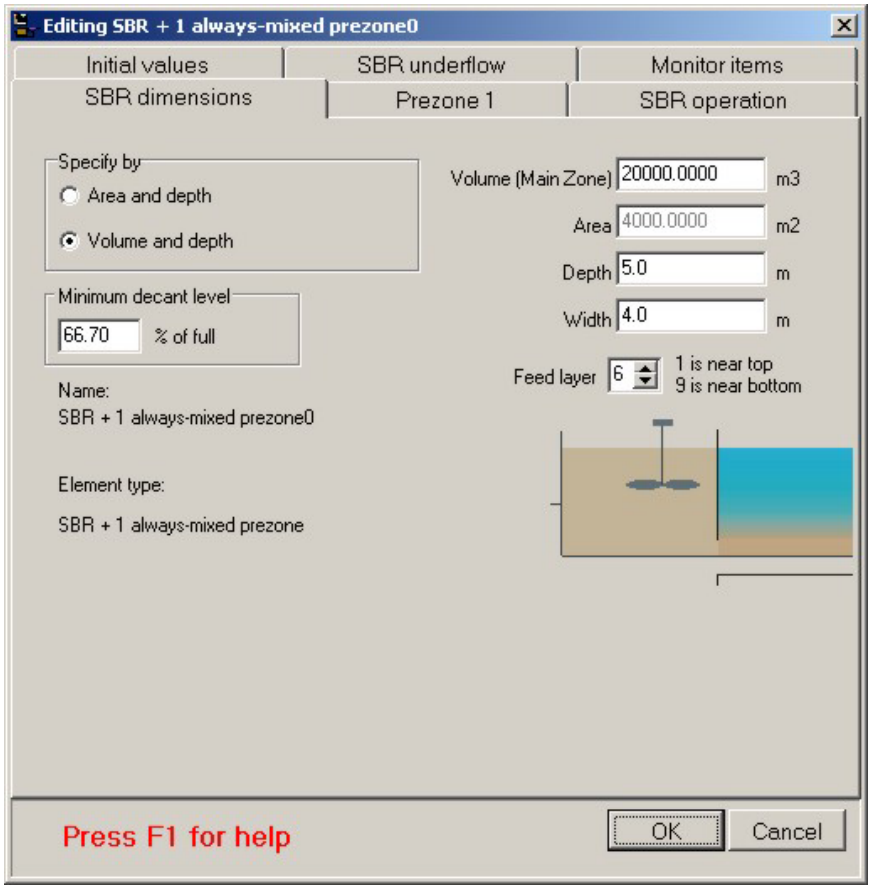
This type of SBR provides one hydraulically linked, always-mixed zone separated from the fill/draw SBR zone by a baffle. You may specify parameters related to the SBR dimensions, cycle operation, starting values, and underflow rates. You also may specify operating conditions for the always-mixed prezone. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The SBR with one always-mixed prezone monitor items tab

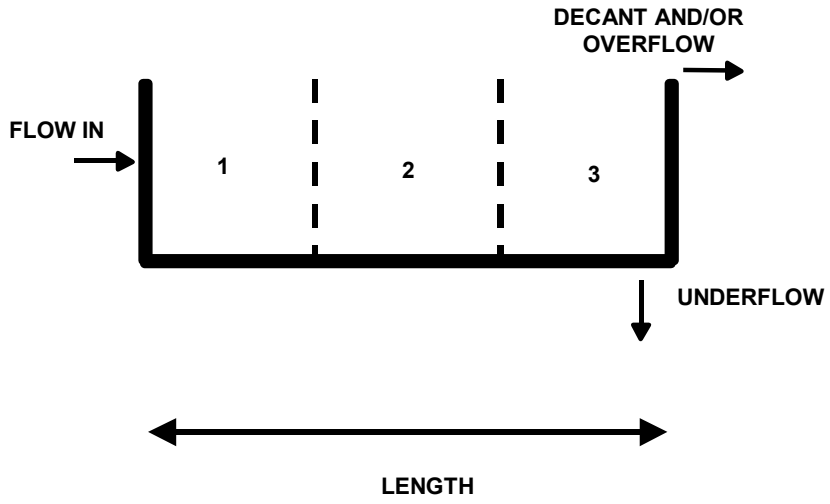
SBR + 1 Always-Mixed Prezone Dimensions

The **SBR Dimensions** tab, shown below, allows the user to enter the physical dimensions of the SBR zone of an SBR + 1 always-mixed prezone element.



The SBR zone dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the SBR zone area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the SBR zone volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Units are shown to the right of the edit boxes. Regardless of the method you choose, you also must specify a **Width** for the SBR zone.



Flow distribution in the SBR zone

BioWin uses the **Width** parameter to calculate the length of the SBR zone. The length is then divided into three equally sized zones. Flow enters the first zone at the level specified by the **Feed layer**. Underflow leaves the SBR zone at the bottom of the third zone, and decant and/or overflow leaves the SBR zone at the top of the third zone.

Some further explanation of how BioWin uses the SBR dimensions is warranted. When you set the dimensions, you specify a **Volume**, **Depth** and **Width** (and **Length** is calculated). Now think of a 2-d side-on view of the SBR with flow from left to right, as shown above.

For example, imagine the full depth is 5 m and the length is 15 m. When settling starts, this is what happens:

- The horizontal length of the SBR is divided into three equal-length subsections (each 5 m in this case).
- The vertical direction is divided into 10 equal-depth layers. [**Note:** as the level goes down during decanting - with no feed - the number of layers stays at 10, and the depth of each layer decreases]. That is, think of each length section as being 10 layers stacked on top of each other (and there are 3 sections).
- Each of the three sections is treated as a vertical settler. That is, the SBR (during settling) is 3 side-by-side settlers, with a total of 30 cells (each completely mixed i.e. of uniform composition).

Consider the dimensions of each of the 30 cells:

- The horizontal area of each cell is **Width * (Length/3) = W * 5 m** here.

- The end area of each cell is **Width * (Liquid Depth/10) = W * 0.5 m** (0.5 m when it is full). [**Note:** in this case the ratio of cell horizontal to side area is 5 : 0.5 = 10 : 1 (if tank is full - but ratio increases when tank level drops)].

For the moment. assume that there is no inflow during decanting. Decant is removed from the top-right cell. During decanting several things happen (apart from settling in each of the 3 sections):

- Say 1 (incremental) unit of volume is removed from the top-right cell (at the composition of the top right cell).
- Overall, the volume of each of the 30 cells must be decreased by 1 unit / 30 (the length of each cell is fixed, so the depth must decrease).
- 29/30ths of the unit volume must get into the top right cell. That is, overall 1/30th of the volume must be removed from each of the other 29 cells.
- The 29/30ths gets into the top right cell from the cell to the left and from the cell below it.
- The cell at the bottom left of the SBR is a special case - there is no flow into it - only outflow of 1/30th. This flow leaves via the top surface and via the right hand end surface. The flow leaving each of the 2 surfaces is proportional to the areas of each surface.
- Flow balances around each of the other 28 cells is handled in a similar manner, doing a simultaneous flow balance on each cell. For the typical cell, there can be:
    - Flow out of the top surface;
    - Flow out the right side;
    - Flow in through the bottom surface; and
    - Flow in through the left surface.

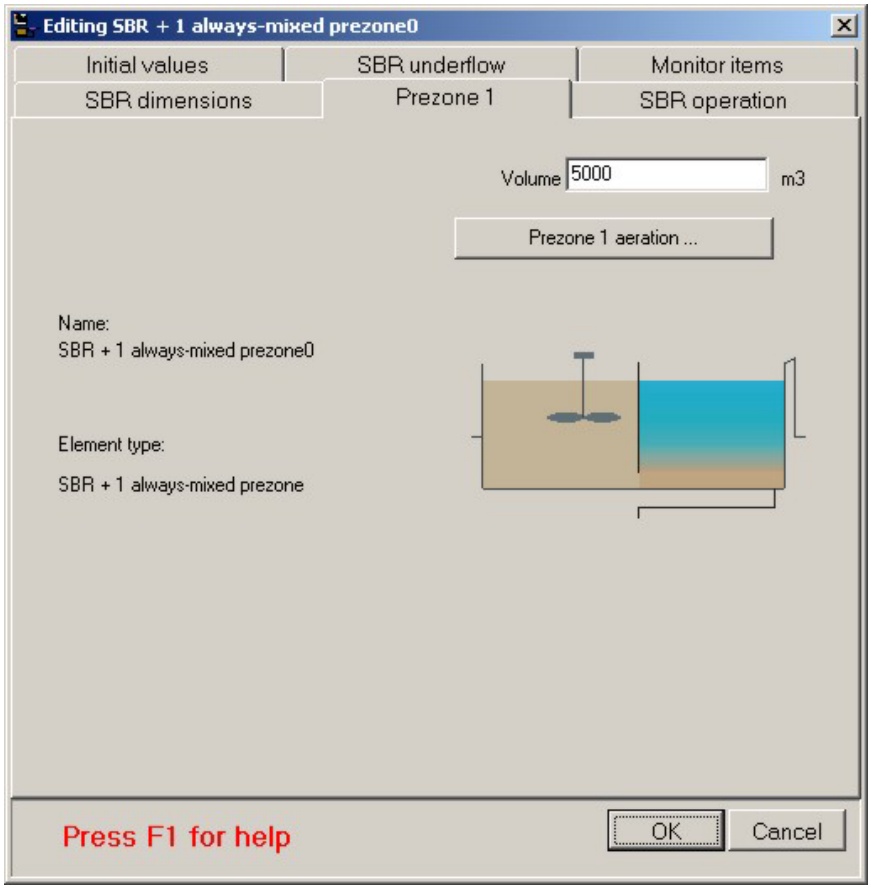
If there is influent flow during decanting, the flow balance is just slightly more complex, but happens in a similar fashion. Note that this explanation makes the process sound like a step-by-step calculation. In reality, what happens is a whole series of simultaneous integrations with error checking.

A **Minimum decant level** must also be specified as a percentage of the height. This level is the lowest liquid level that can be achieved by decanting liquid out of the top of the SBR zone. Note that it *is* possible to obtain lower liquid levels in a cycle than this value, but to get lower than this level, the liquid would have to go out via the underflow.

The **Feed layer** also is specified on this tab. This is the layer where liquid flow from the always mixed prezone is introduced to the main SBR zone. Note that the layers are numbered such that layer 1 is near the top, and number 9 is near the bottom.

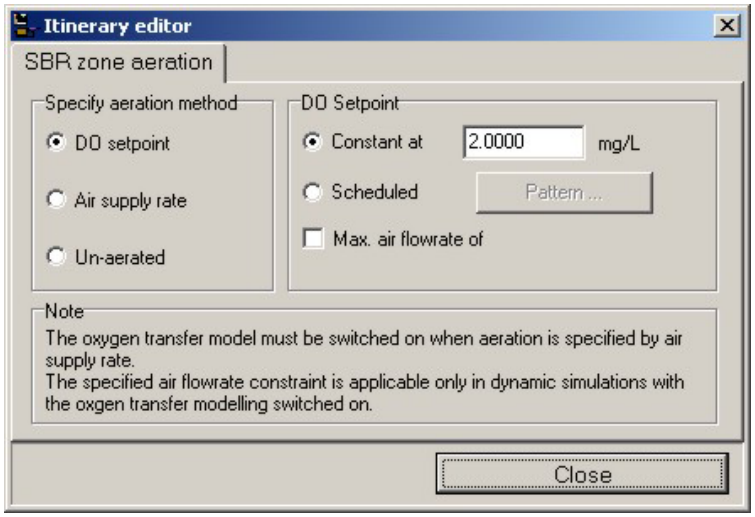
Single Always-Mixed Prezone Setup

The **Prezone 1** tab shown below is used for specifying the volume and aeration settings of the prezone for the SBR + 1 always-mixed prezone element.



The single always-mixed prezone configuration tab

You can specify the volume of the prezone using the **Volume** text edit field. You can set up the prezone aeration operation by clicking the **Prezone 1 aeration…** button. Doing so will present you with the dialog box shown below.



The single always-mixed prezone aeration settings dialog box

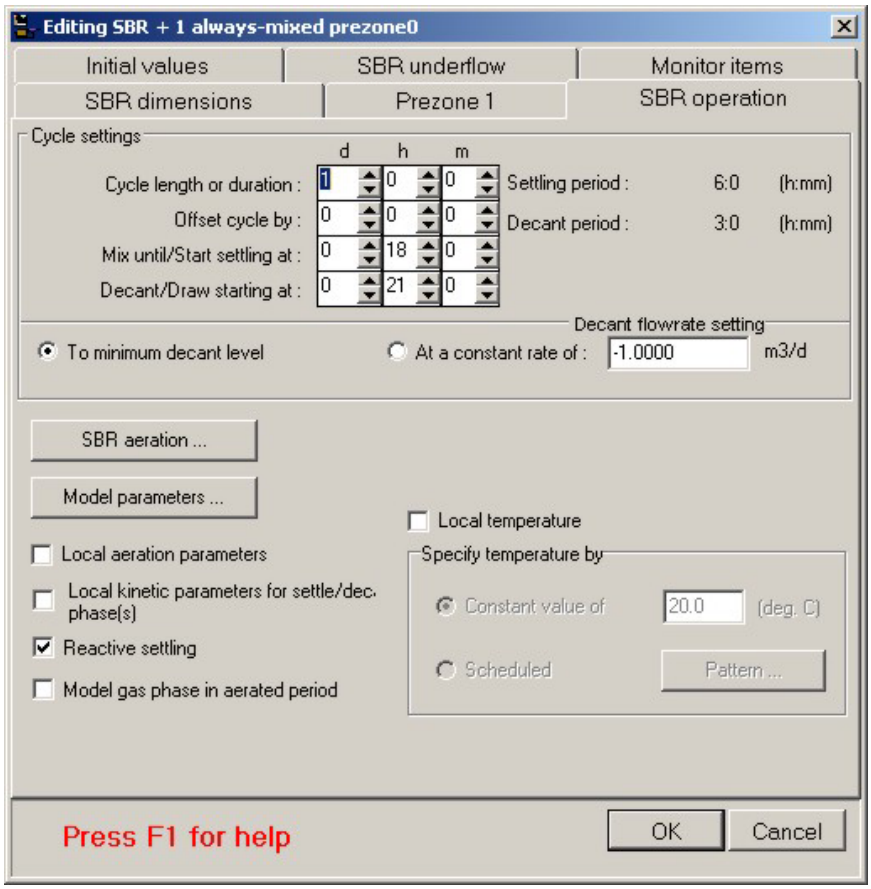
There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be

specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **Edit DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **Edit air flow itinerary** dialog box).

SBR + 1 Always-Mixed Prezone SBR Zone Operation

The **SBR Operation** tab, shown below, allows the user to enter operating parameters for the SBR zone of an SBR + 1 always-mixed prezone element, such as cycle patterns, decant flow rates, aeration specifications and local temperature, as well as change local model parameters.



The SBR zone operation tab

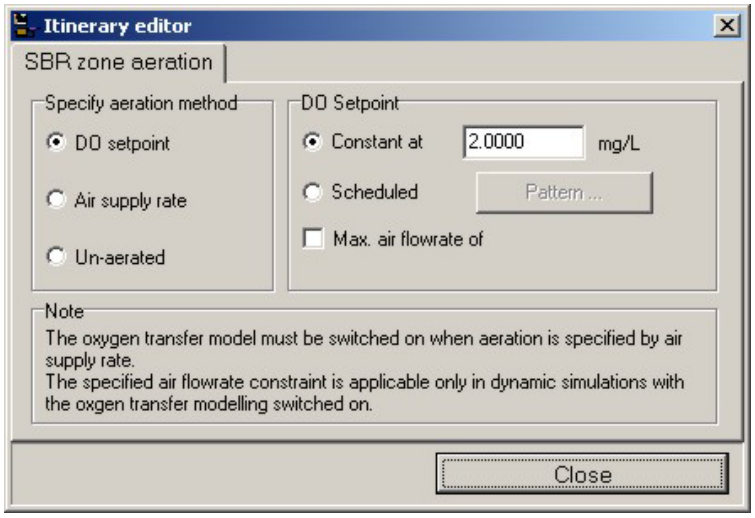
The major operational settings of the SBR zone of the SBR + 1 always-mixed prezone are those related to **Cycle settings**. The **Cycle length or duration** is the total time that will elapse before the cycle begins to repeat itself. The first event to take place in the cycle is the end of the mixing phase/beginning of the settling phase. You specify this event time with the **Mix until/Start settling at:** spin edit control. Note that the dialog provides you with feedback as to the length of your settling period The second event to take place is the beginning of the decant phase. You specify this event time with the **Decant/Draw starting at:** spin edit control. Note that the dialog provides you with feedback as to the length of your decant

period. Experimenting with these settings will reveal that settling and decanting take place simultaneously – but the settling phase length always must be greater than the decant phase length. Finally, you may set up a cycle offset if you wish using the **Offset cycle by:** control.

Note that BioWin assumes that there is no influent or underflow activity during the decant period and bases the calculation for required decant flowrate on that premise.

The next operational setting is the **Decant flowrate setting**. You may specify this with one of two possible methods. If you know the flowrate you wish to decant at, then choose the **At a constant rate of:** option and enter the desired value. If you wish BioWin to calculate the decant flowrate for you, select the **To minimum decant level** option. The decant flowrate is calculated as follows. At the beginning of the decant period, the simulator looks at the difference between the current liquid volume and the specified minimum decant volume. It then calculates the necessary flowrate to decant the available liquid during the decant period.

You can specify the SBR zone aeration settings by clicking the **SBR aeration…** button which will show the following dialog box.



The SBR zone aeration setting dialog box

There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **SBR DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **SBR air flow itinerary** dialog box.

Note that if you wish to change any of the aeration model parameters, click the **Model parameters…** button in the main **Operation** tab to open the **Model parameter editor** dialog box (Note that this changes the aeration model parameters on a local level only).

A **Local temperature** also may be specified for an SBR + 1 always-mixed prezone element. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may

enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

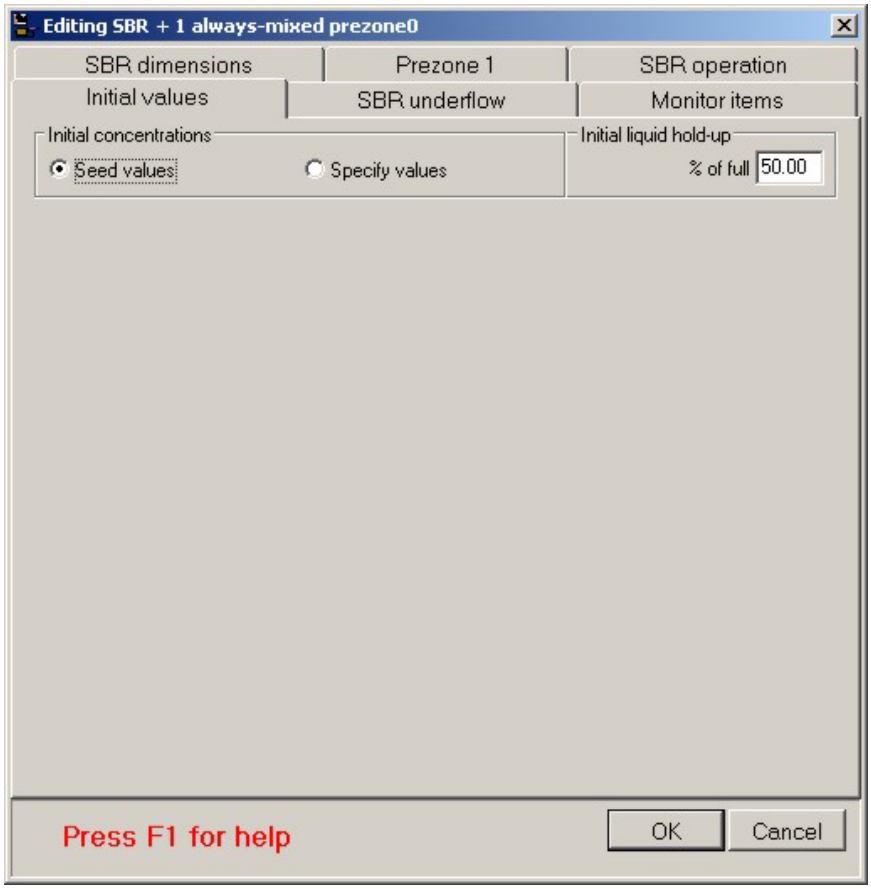
You also can specify **Local kinetic parameters** for the activated sludge model used in the SBR zone **decant/settling phase**. For the mixing phase (and for the prezone), the global model parameter values are used. If you click on the check box for local kinetic parameters, then clicking the **Model parameters…** button opens the **Model parameter editor** dialog box allowing you access to the various activated sludge model kinetic parameters.

To specify local aeration parameters (e.g. alpha and beta factors), check the box labeled **Local aeration parameters**.

To model the concentrations of the constituents of the aeration gas (e.g. oxygen, carbon dioxide, etc.), check the box labeled **Model gas phase**. If this option is selected you may also enter a percentage value for the **Gas hold-up** (i.e. the percentage of the total reactor volume occupied by aeration gas).

SBR + 1 Always-Mixed Prezone Initial Values

The **Initial values** tab, shown below, is used for setting up the initial settings for SBR + 1 always-mixed prezone concentrations and volume.



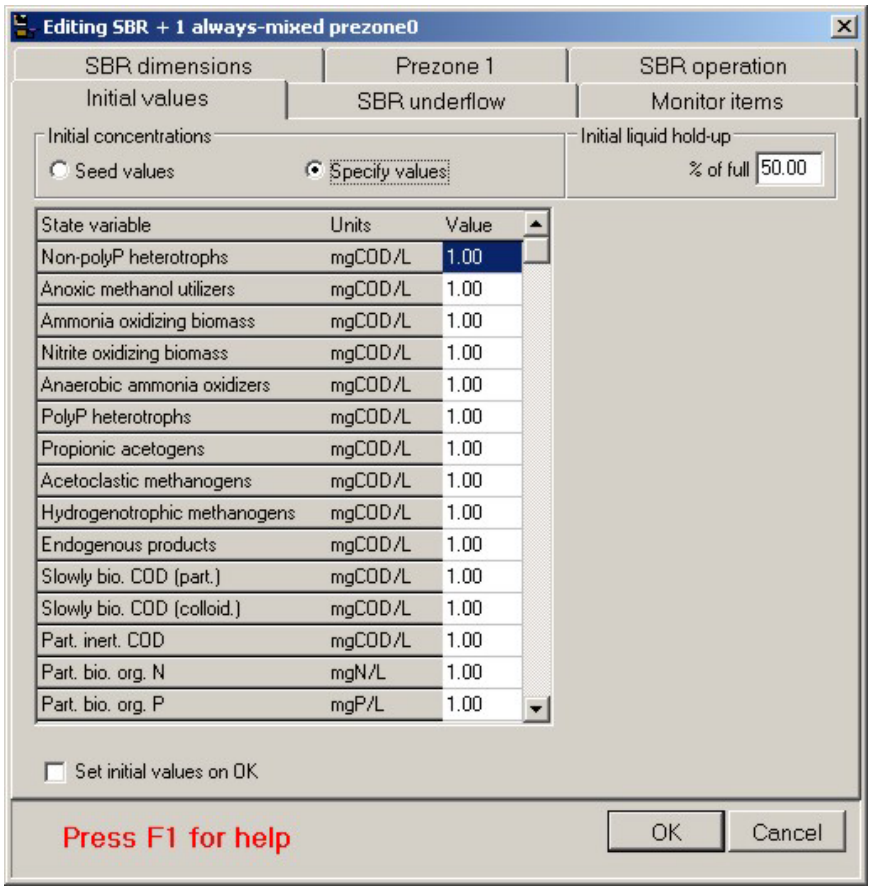
The SBR + 1 always-mixed prezone initial values tab with the Seed values *option selected*

Use the **Initial liquid hold-up** field to enter a value to specify the **% of full** setting. It should be noted here that the lower and upper limits on this value are 0.02 and 99.98%, respectively. This reflects the fact that this is the liquid volume, and allows

for small differences between *liquid* volume and *reactor* volume for inlet and outlet pipes.

There are two methods for setting up **Initial concentrations** in the SBR + 1 always-mixed prezone. The dialog box shown above illustrates the first case where the **Seed values** option is selected. When this option is selected, BioWin applies default seed values for all the state variables (except volume, which you specify) in the same manner it would for other bioreactor elements *in all of the zones* (i.e. including the prezone).

The dialog box shown below illustrates the second case where the **Specify values** option is selected. In this case, an editable list of state variables is displayed in the dialog box. This allows you to enter your own seed values for state variables in the SBR + 1 always-mixed prezone. These will be inserted in each zone of the SBR + 1 always-mixed prezone element when you begin a dynamic simulation and choose to use seed values.



The SBR + 1 always-mixed prezone initial values tab with the Specify values *option selected*

When the specified seed values get inserted in the SBR + 1 always-mixed prezone is determined by the **Set initial values on OK** option. If this option **is not** checked:

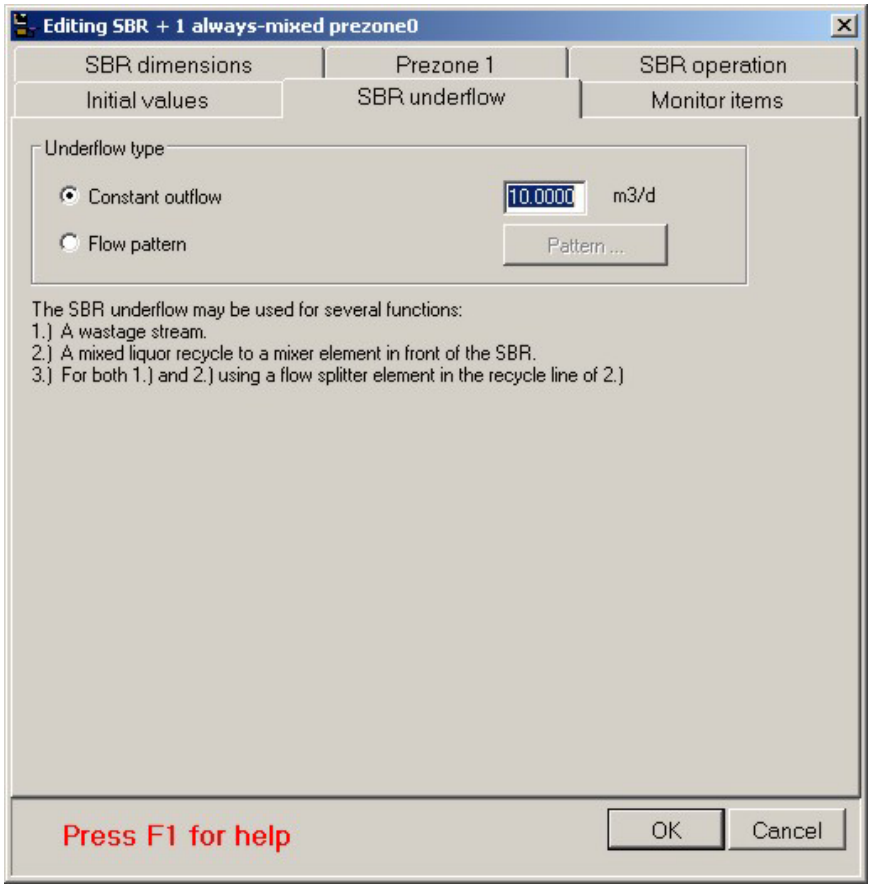
- The specified values will be placed in the SBR + 1 always-mixed prezone at the beginning of a dynamic simulation **if** you elect to seed the simulation.

If this option **is** checked, the specified values are placed in the SBR + 1 always-mixed prezone when you click **OK** to close the dialog box, **overriding any**

existing state variable values. Therefore, the state variable values at the beginning of a dynamic simulation will be the same regardless of whether or not you choose to re-seed.

SBR + 1 Always-Mixed Prezone Underflow

The SBR + 1 always-mixed prezone element **SBR Underflow** tab shown below allows you to set up underflow rates and patterns for the SBR zone.



The SBR zone underflow settings tab

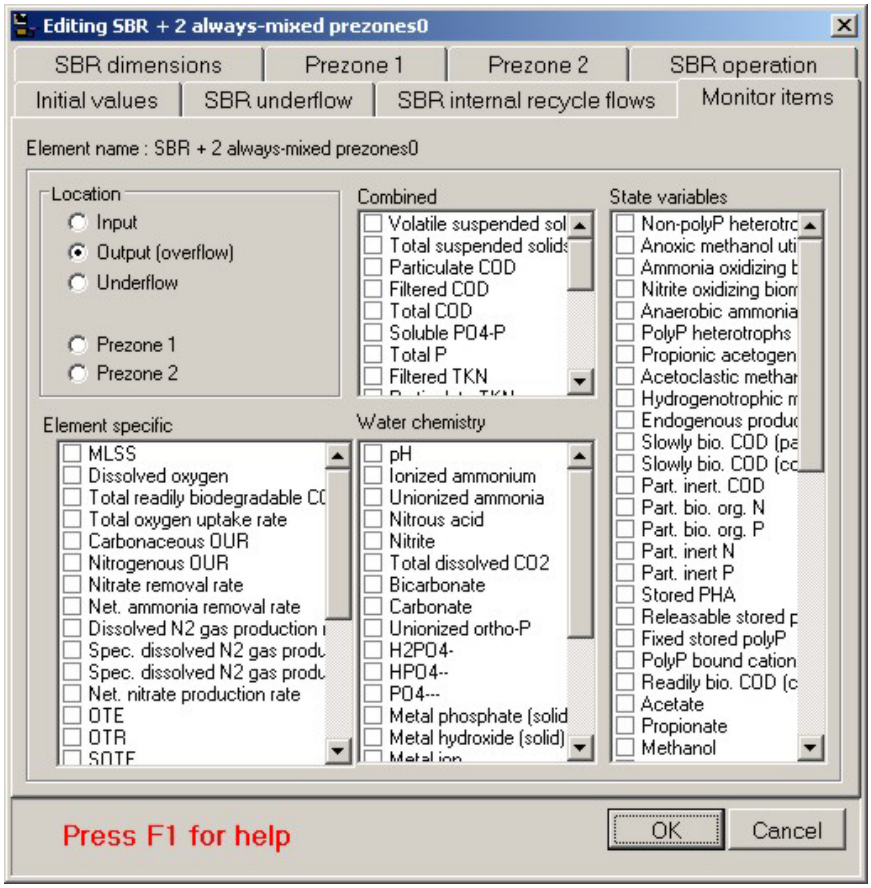
The **Underflow type** may be specified as a **Constant underflow** or a **Flow pattern** by selecting the appropriate option. If you select **Constant underflow**, you may enter the desired underflow rate in the edit field. If you select **Flow pattern**, clicking the **Pattern…** button will open the **Underflow rate itinerary** dialog box.

The SBR zone underflow may be used for several functions:

1. As a wastage stream;
2. A mixed liquor recycle to a mixer element in front of the SBR + 1 always-mixed prezone element;
3. As both a wastage and a recycle by placing a splitter element between the underflow and the mixer element in front of the SBR + 1 always-mixed prezone element.

SBR with Two Always-Mixed Prezones

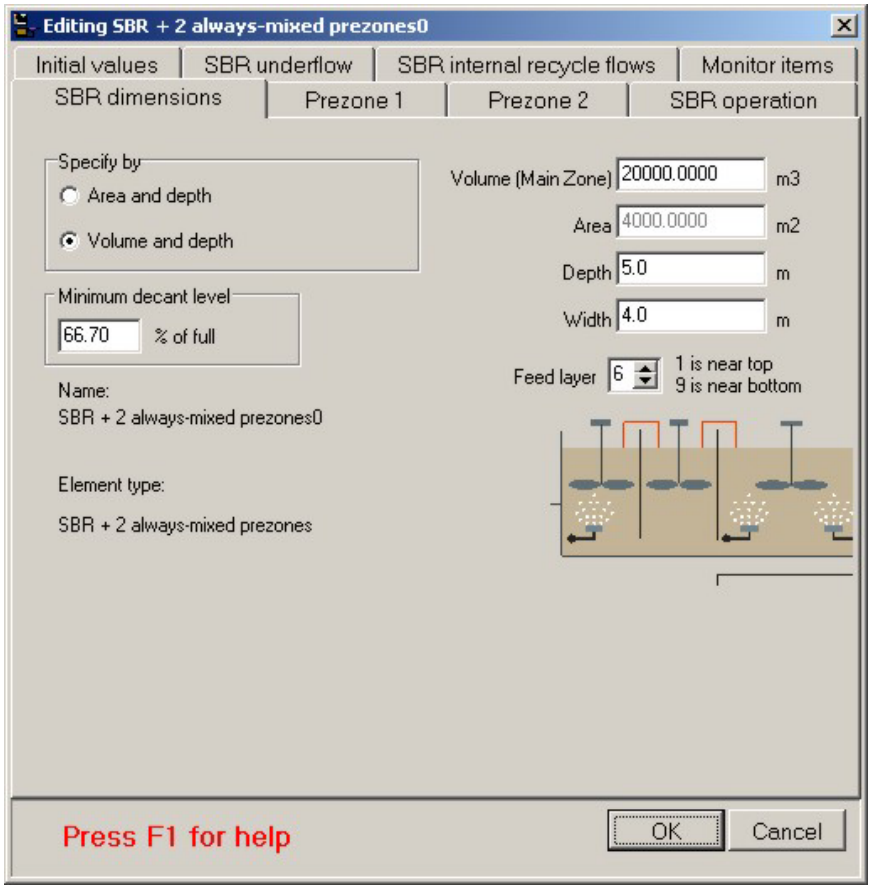
This type of SBR provides two hydraulically linked, always-mixed zones separated from the fill/draw SBR zone by a baffle. You may specify parameters related to the SBR dimensions, cycle operation, starting values, and underflow rates. You also may specify operating conditions for the always-mixed prezone. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The SBR + 2 always-mixed prezones monitor items tab

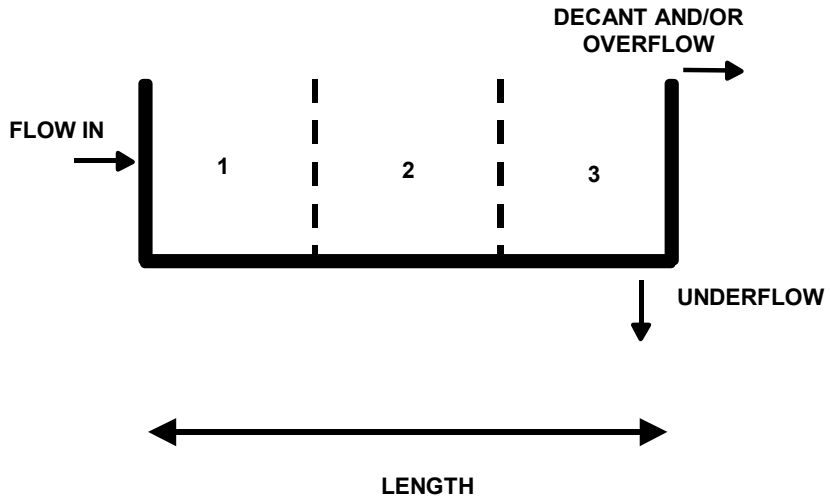
SBR + 2 Always-Mixed Prezones Dimensions

The **SBR Dimensions** tab, shown below, allows the user to enter the physical dimensions of the SBR zone of an SBR + 2 always-mixed prezones element.



The SBR + 2 always-mixed prezones dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the SBR zone area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the SBR zone volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Units are shown to the right of the edit boxes. Regardless of the method you choose, you also must specify a **Width** for the SBR zone.



Flow distribution in the SBR zone

BioWin uses the **Width** parameter to calculate the length of the SBR zone. The length is then divided into three equally sized zones. Flow enters the first zone at the level specified by the **Feed layer**. Underflow leaves the SBR zone at the bottom of the third zone, and decant and/or overflow leaves the SBR zone at the top of the third zone.

Some further explanation of how BioWin uses the SBR dimensions is warranted. When you set the dimensions, you specify a **Volume**, **Depth** and **Width** (and **Length** is calculated). Now think of a 2-d side-on view of the SBR with flow from left to right, as shown above.

For example, imagine the full depth is 5 m and the length is 15 m. When settling starts, this is what happens:

- The horizontal length of the SBR is divided into three equal-length subsections (each 5 m in this case).
- The vertical direction is divided into 10 equal-depth layers. [**Note:** as the level goes down during decanting - with no feed - the number of layers stays at 10, and the depth of each layer decreases]. That is, think of each length section as being 10 layers stacked on top of each other (and there are 3 sections).
- Each of the three sections is treated as a vertical settler. That is, the SBR (during settling) is 3 side-by-side settlers, with a total of 30 cells (each completely mixed i.e. of uniform composition).

Consider the dimensions of each of the 30 cells:

- The horizontal area of each cell is **Width * (Length/3) = W * 5 m** here.

- The end area of each cell is **Width * (Liquid Depth/10) = W * 0.5 m** (0.5 m when it is full). [**Note:** in this case the ratio of cell horizontal to side area is 5 : 0.5 = 10 : 1 (if tank is full - but ratio increases when tank level drops)].

For the moment. assume that there is no inflow during decanting. Decant is removed from the top-right cell. During decanting several things happen (apart from settling in each of the 3 sections):

- Say 1 (incremental) unit of volume is removed from the top-right cell (at the composition of the top right cell).
- Overall, the volume of each of the 30 cells must be decreased by 1 unit / 30 (the length of each cell is fixed, so the depth must decrease).
- 29/30ths of the unit volume must get into the top right cell. That is, overall 1/30th of the volume must be removed from each of the other 29 cells.
- The 29/30ths gets into the top right cell from the cell to the left and from the cell below it.
- The cell at the bottom left of the SBR is a special case - there is no flow into it - only outflow of 1/30th. This flow leaves via the top surface and via the right hand end surface. The flow leaving each of the 2 surfaces is proportional to the areas of each surface.
- Flow balances around each of the other 28 cells is handled in a similar manner, doing a simultaneous flow balance on each cell. For the typical cell, there can be:
    - Flow out of the top surface;
    - Flow out the right side;
    - Flow in through the bottom surface; and
    - Flow in through the left surface.

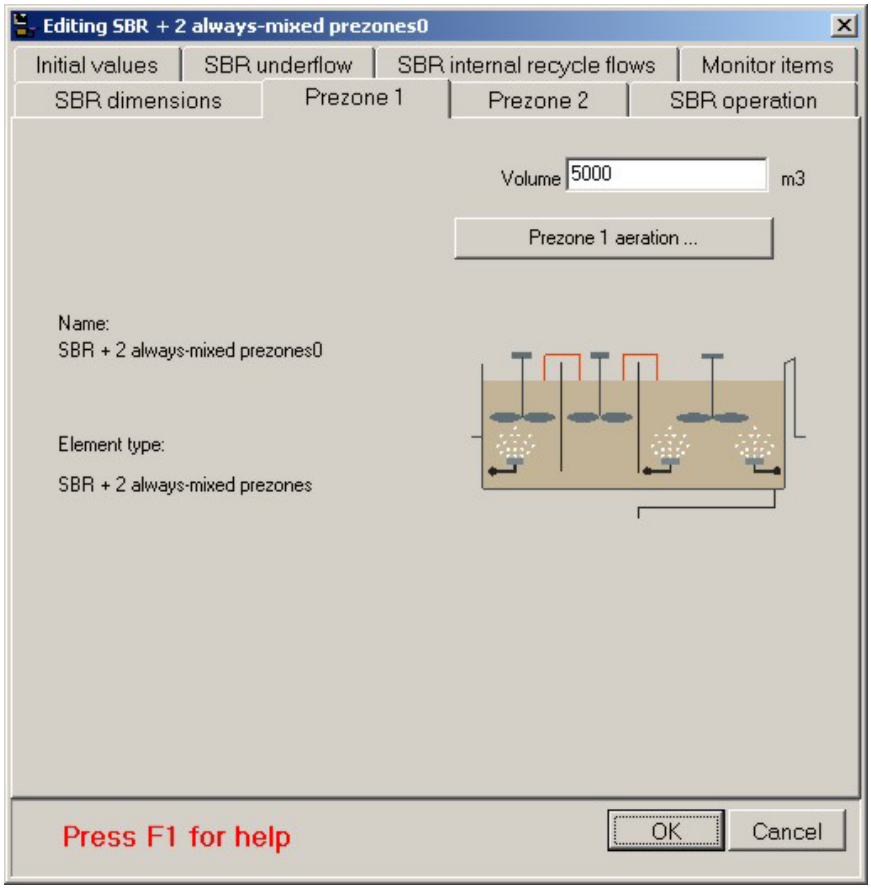
If there is influent flow during decanting, the flow balance is just slightly more complex, but happens in a similar fashion. Note that this explanation makes the process sound like a step-by-step calculation. In reality, what happens is a whole series of simultaneous integrations with error checking.

A **Minimum decant level** must also be specified as a percentage of the height. This level is the lowest liquid level that can be achieved by decanting liquid out of the top of the SBR zone. Note that it *is* possible to obtain lower liquid levels in a cycle than this value, but to get lower than this level, the liquid would have to go out via the underflow.

The **Feed layer** also is specified on this tab. This is the layer where liquid flow is introduced to the element. Note that the layers are numbered such that layer 1 is near the top, and number 9 is near the bottom. The feed flow is distributed throughout the SBR + 2 always-mixed prezones element according to the vessel geometry.

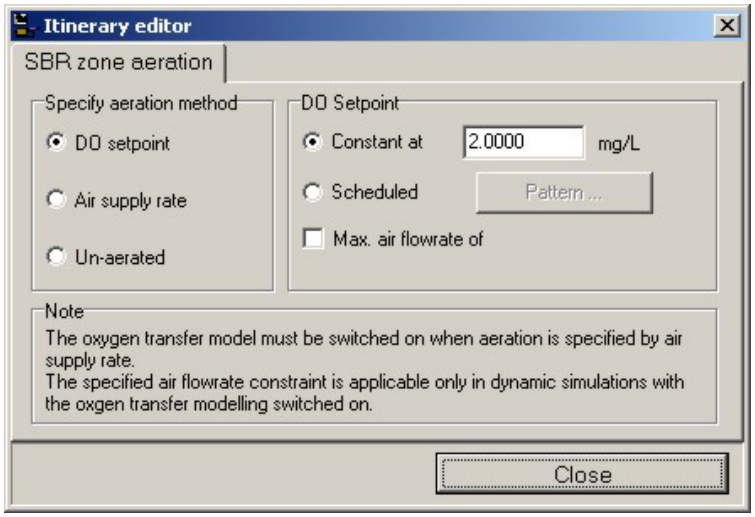
First Always-Mixed Prezone Setup

The **Prezone 1** tab shown below is used for specifying the volume and aeration settings of the first prezone for the SBR + 2 always-mixed prezones element.



The first always mixed prezone configuration tab

You can specify the volume of the first prezone using the **Volume** text edit field. You can set up the first prezone aeration operation by clicking the **Prezone 1 aeration…** button. Doing so will present you with the dialog box shown below.



The first prezone aeration settings dialog box

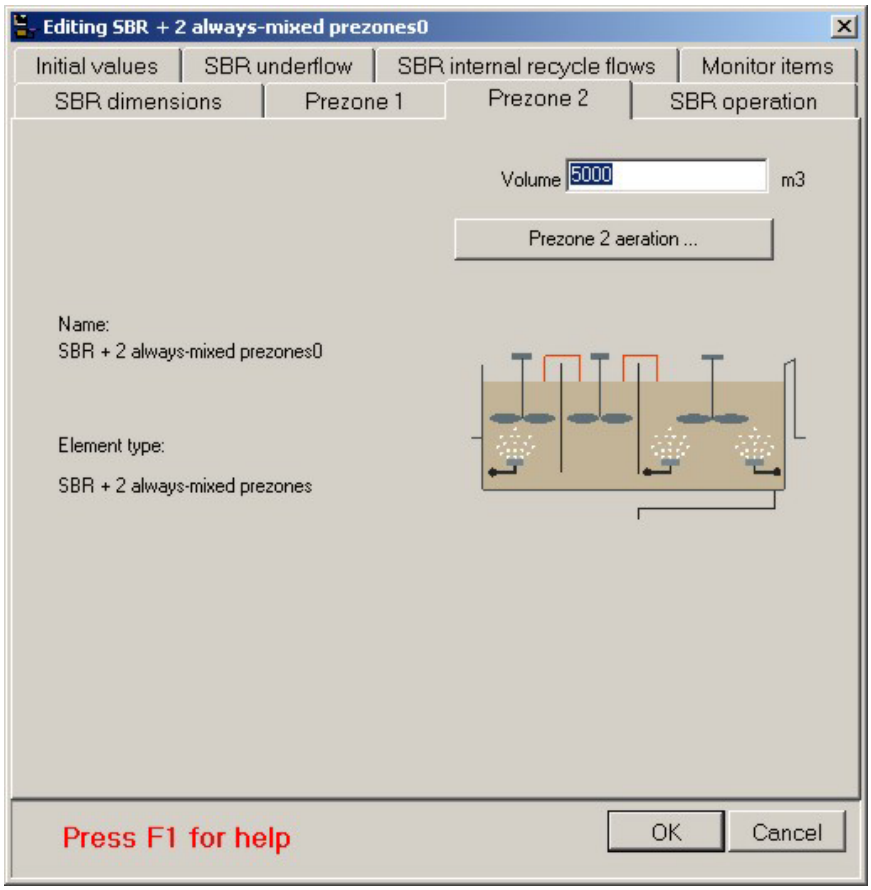
There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be

specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **Edit DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **Edit air flow itinerary** dialog box).

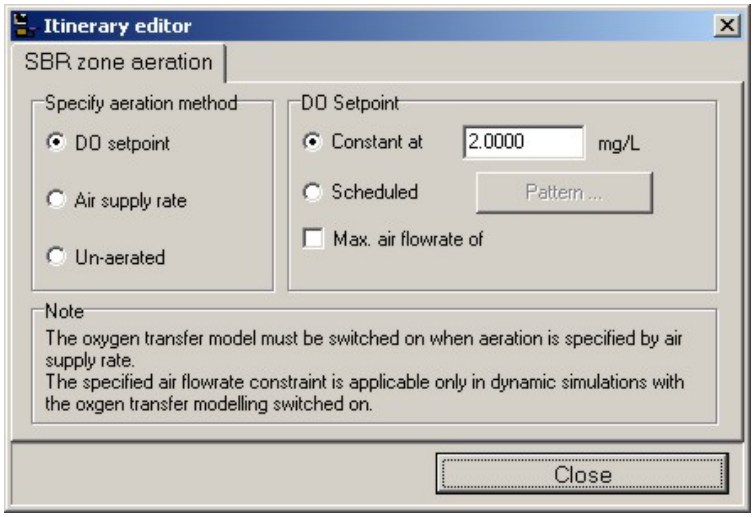
Second Always-Mixed Prezone Setup

The **Prezone 2** tab shown below is used for specifying the volume and aeration settings of the second prezone for the SBR + 2 always-mixed prezones element.



The second always mixed prezone configuration tab

You can specify the volume of the second prezone using the **Volume** text edit field. You can set up the second prezone aeration operation by clicking the **Prezone 2 aeration…** button. Doing so will present you with the dialog box shown below.



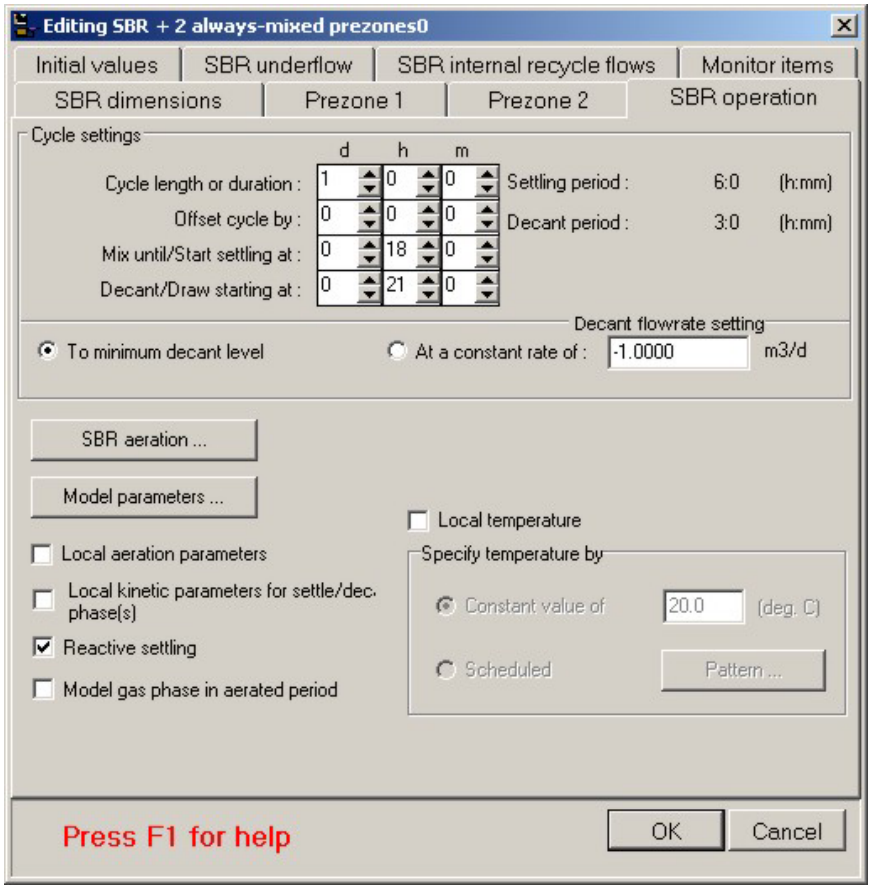
The second prezone aeration settings dialog box

There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **Edit DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **Edit air flow itinerary** dialog box).

SBR + 2 Always-Mixed Prezones SBR Zone Operation

The **SBR Operation** tab, shown below, allows the user to enter operating parameters for the SBR zone of an SBR + 2 always-mixed prezones element, such as cycle patterns, decant flow rates, aeration specifications and local temperature, as well as change local model parameters.



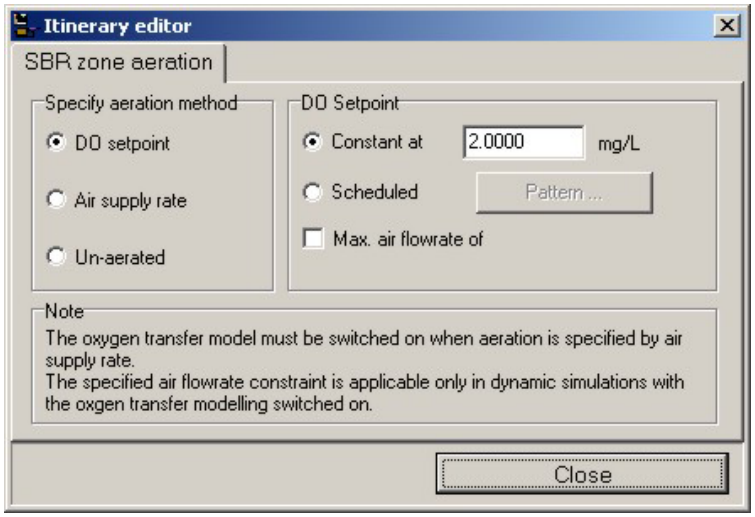
The SBR + 2 always-mixed prezones operation tab

The major operational settings of the SBR zone of the SBR + 2 always-mixed prezones are those related to **Cycle settings**. The **Cycle length or duration** is the total time that will elapse before the cycle begins to repeat itself. The first event to take place in the cycle is the end of the mixing phase/beginning of the settling phase. You specify this event time with the **Mix until/Start settling at:** spin edit control. Note that the dialog provides you with feedback as to the length of your settling period The second event to take place is the beginning of the decant phase. You specify this event time with the **Decant/Draw starting at:** spin edit control. Note that the dialog provides you with feedback as to the length of your decant period. Experimenting with these settings will reveal that settling and decanting take place simultaneously – but the settling phase length always must be greater than the decant phase length. Finally, you may set up a cycle offset if you wish using the **Offset cycle by:** control.

Note that BioWin assumes that there is no influent or underflow activity during the decant period and bases the calculation for required decant flowrate on that premise.

The next operational setting is the **Decant flowrate setting**. You may specify this with one of two possible methods. If you know the flowrate you wish to decant at, then choose the **At a constant rate of:** option and enter the desired value. If you wish BioWin to calculate the decant flowrate for you, select the **To minimum decant level** option. The decant flowrate is calculated as follows. At the beginning of the decant period, the simulator looks at the difference between the current liquid volume and the specified minimum decant volume. It then calculates the necessary flowrate to decant the available liquid during the decant period.

You can specify the SBR zone aeration settings by clicking the **SBR aeration…** button which will show the following dialog box.



The SBR zone aeration setting dialog box

There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **SBR DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **SBR air flow itinerary** dialog box).

Note that if you wish to change any of the aeration model parameters, click the **Model parameters…** button in the main **Operation** tab to open the **Model parameter editor** dialog box (Note that this changes the aeration model parameters on a local level only).

A **Local temperature** also may be specified for an SBR + 2 always-mixed prezones element. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

You also can specify **Local kinetic parameters** for the activated sludge model used in the SBR zone **decant/settling phase**. For the mixing phase (and for the prezones), the global model parameter values are used. If you click on the check box for local kinetic parameters, then clicking the **Model parameters…** button opens the **Model parameter editor** dialog box allowing you access to the various activated sludge model kinetic parameters.

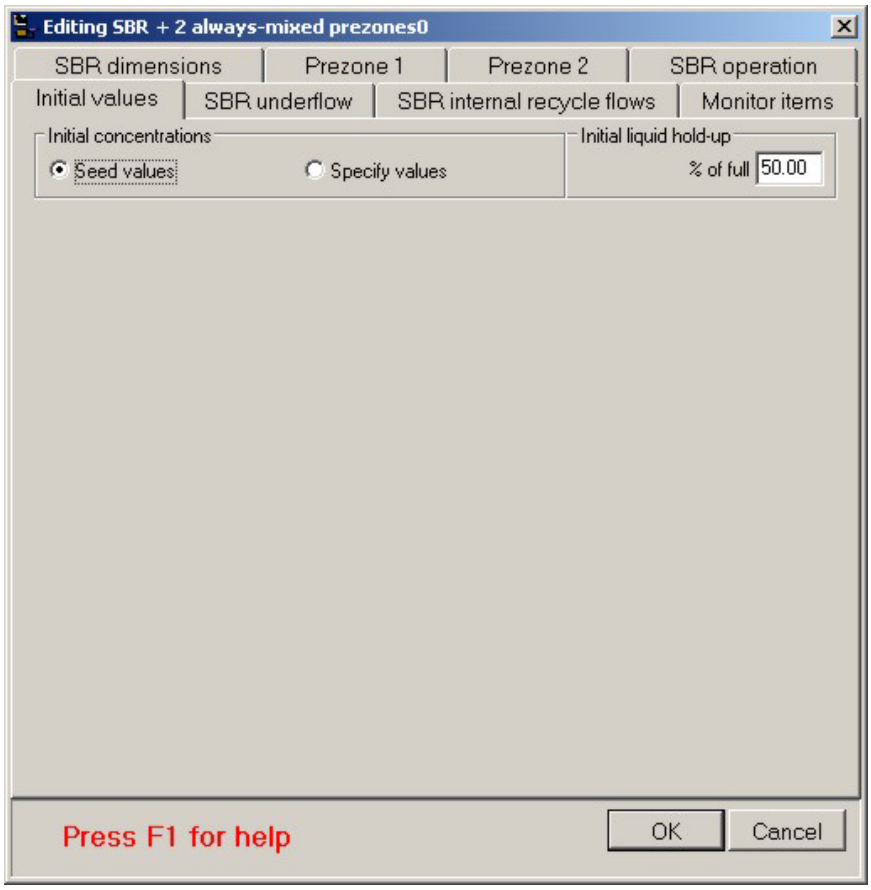
To specify local aeration parameters (e.g. alpha and beta factors), check the box labeled **Local aeration parameters**.

To model the concentrations of the constituents of the aeration gas (e.g. oxygen, carbon dioxide, etc.), check the box labeled **Model gas phase**. If this option is

selected you may also enter a percentage value for the **Gas hold-up** (i.e. the percentage of the total reactor volume occupied by aeration gas).

SBR + 2 Always-Mixed Prezones Initial Values

The **Initial values** tab, shown below, is used for setting up the initial settings for SBR + 2 always-mixed prezones concentrations and volume.

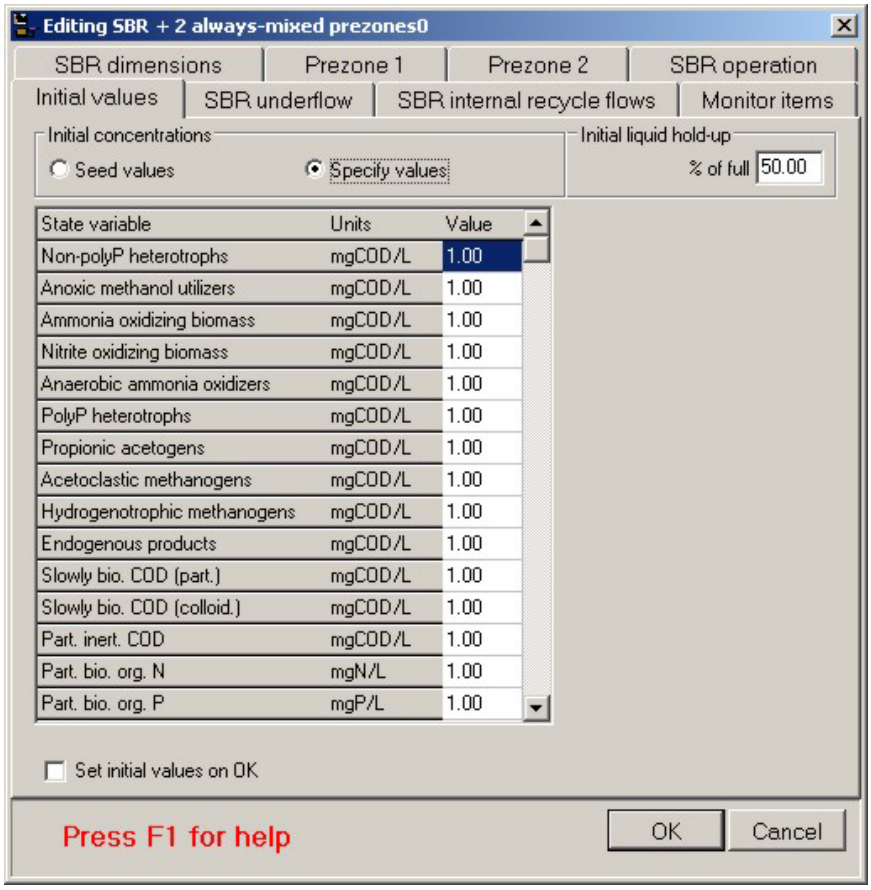


The SBR + 2 always-mixed prezones initial values tab with the Seed values *option selected*

Use the **Initial liquid hold-up** field to enter a value to specify the **% of full** setting. It should be noted here that the lower and upper limits on this value are 0.02 and 99.98%, respectively. This reflects the fact that this is the liquid volume, and allows for small differences between *liquid* volume and *reactor* volume for inlet and outlet pipes.

There are two methods for setting up **Initial concentrations** in the SBR + 2 always-mixed prezones. The dialog box shown above illustrates the first case where the **Seed values** option is selected. When this option is selected, BioWin applies default seed values for all the state variables (except volume, which you specify) in the same manner it would for other bioreactor elements *in all of the zones* (i.e. including the prezones).

The dialog box shown below illustrates the second case where the **Specify values** option is selected. In this case, an editable list of state variables is displayed in the dialog box. This allows you to enter your own seed values for state variables in the SBR + 2 always-mixed prezones element. These will be inserted in each zone of the SBR + 2 always-mixed prezones element when you begin a dynamic simulation and choose to use seed values.



The SBR + 2 always-mixed prezones initial values tab with the Specify values *option selected*

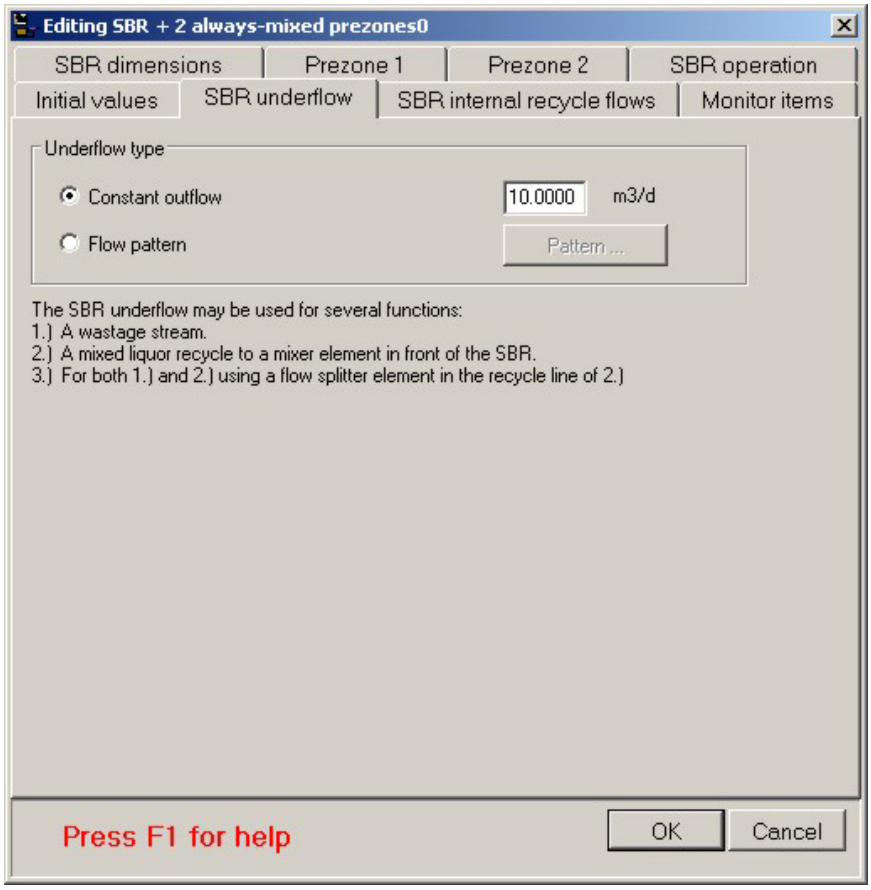
When the specified seed values get inserted in the SBR + 2 always-mixed prezones is determined by the **Set initial values on OK** option. If this option **is not** checked:

- The specified values will be placed in the SBR + 2 always-mixed prezones at the beginning of a dynamic simulation **if** you elect to seed the simulation.

If this option **is** checked, the specified values are placed in the SBR + 2 always-mixed prezones when you click **OK** to close the dialog box, **overriding any existing state variable values**. Therefore, the state variable values at the beginning of a dynamic simulation will be the same regardless of whether or not you choose to re-seed.

SBR + 2 Always-Mixed Prezones Underflow

The SBR + 2 always-mixed prezones **SBR Underflow** tab shown below allows you to set up underflow rates and patterns for the SBR zone.



The SBR zone underflow settings tab

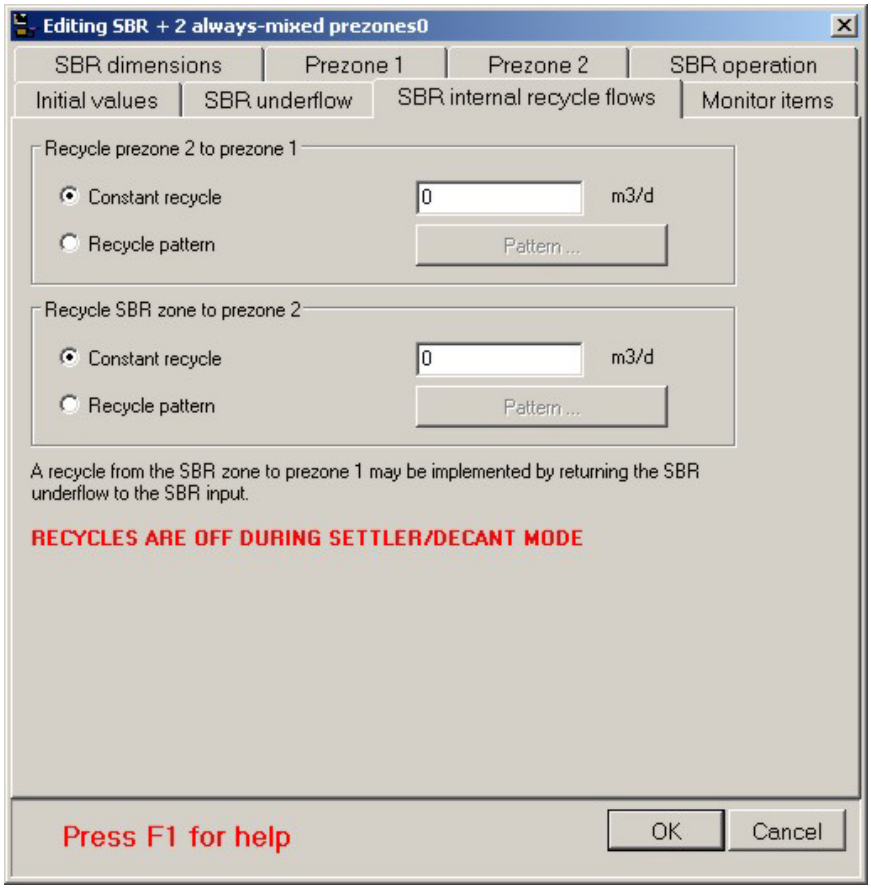
The **Underflow type** may be specified as a **Constant underflow** or a **Flow pattern** by selecting the appropriate option. If you select **Constant underflow**, you may enter the desired underflow rate in the edit field. If you select **Flow pattern**, clicking the **Pattern…** button will open the **Underflow rate itinerary** dialog box.

The SBR zone underflow may be used for several functions:

1. As a wastage stream;
2. A mixed liquor recycle to a mixer element in front of the SBR + 2 always-mixed prezones element;
3. As both a wastage and a recycle by placing a splitter element between the underflow and the mixer element in front of the SBR + 2 always-mixed prezones element.

SBR + 2 Always-Mixed Prezones Internal Recycle Flows

The SBR + 2 always-mixed prezones element **SBR internal recycle flows** tab shown below allows you to set up recycles between the SBR zone and the second prezone, and between the second and first prezone.



The SBR internal recycle flows tab

You may have a recycle flow which goes from the second prezone to the first one. If you select **Constant recycle**, you may enter the desired recycle flowrate in the edit field. If you select **Recycle pattern**, clicking the **Pattern…** button will open the **Recycle flowrate itinerary** dialog box.

You also may have a recycle flow which goes from the SBR zone to the second prezone. If you select **Constant recycle**, you may enter the desired recycle flowrate in the edit field. If you select **Recycle pattern**, clicking the **Pattern…** button will open the **Recycle flowrate itinerary** dialog box.

Two final points should be noted with regard to internal recycle flows:

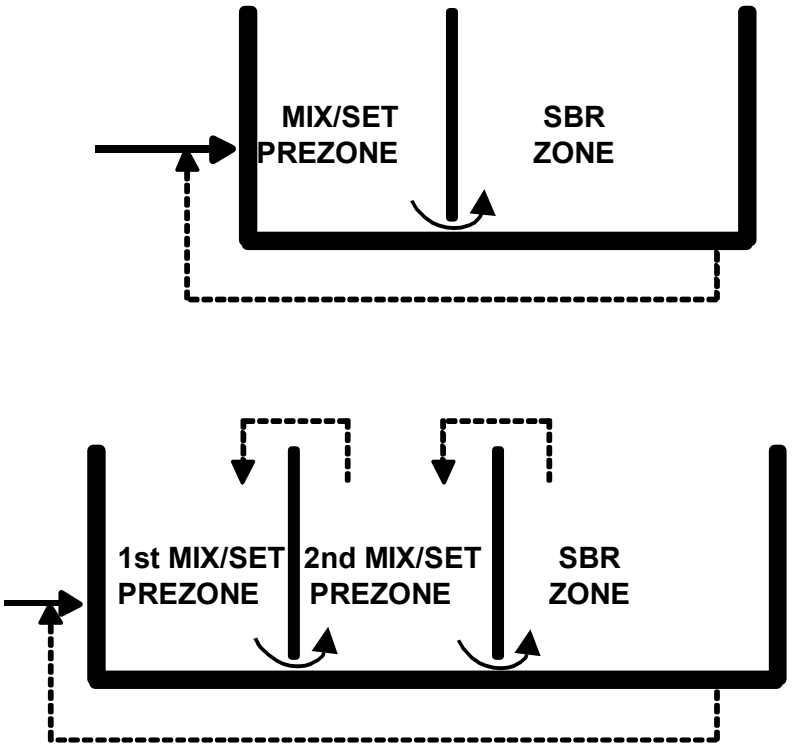
1. It is possible to have a recycle flow from the SBR zone to the first prezone by returning the SBR zone underflow (or a portion of it) to the input of the SBR + 2 always-mixed prezones element.
2. Internal recycle flows are set to zero when the SBR zone goes into settle/decant mode.

SBR with Mix/Settle Prezone(s)

This type of SBR allows you to have hydraulically-linked prezones that go into settling mode when the decant zone goes into settle mode. The following SBR with mix/settle prezone combinations are available:

- SBR + 1 Prezone
- SBR + 2 Prezones

BioWin allows you to have internal recycles between the prezones and the SBR, as shown in the following diagram.

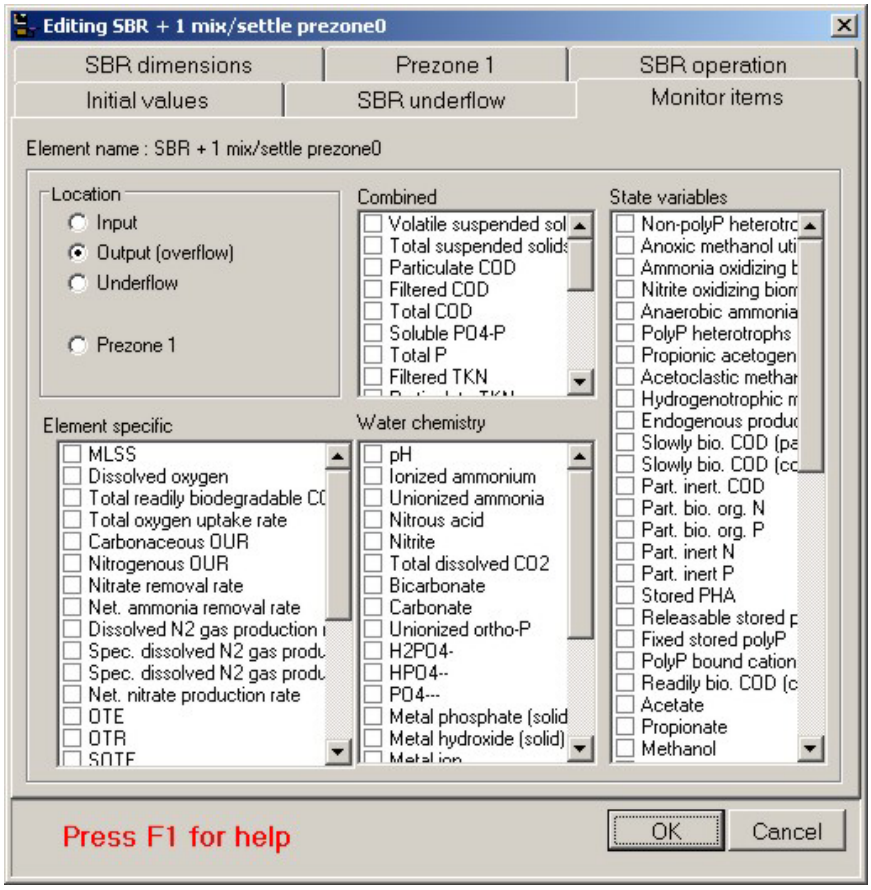


Flow recycles and patterns that can be set up between an SBR and its prezone(s)

The following sections describe the dialog boxes for the SBR with one and two mix/settle prezone(s), respectively.

SBR with One Mix/Settle Prezone

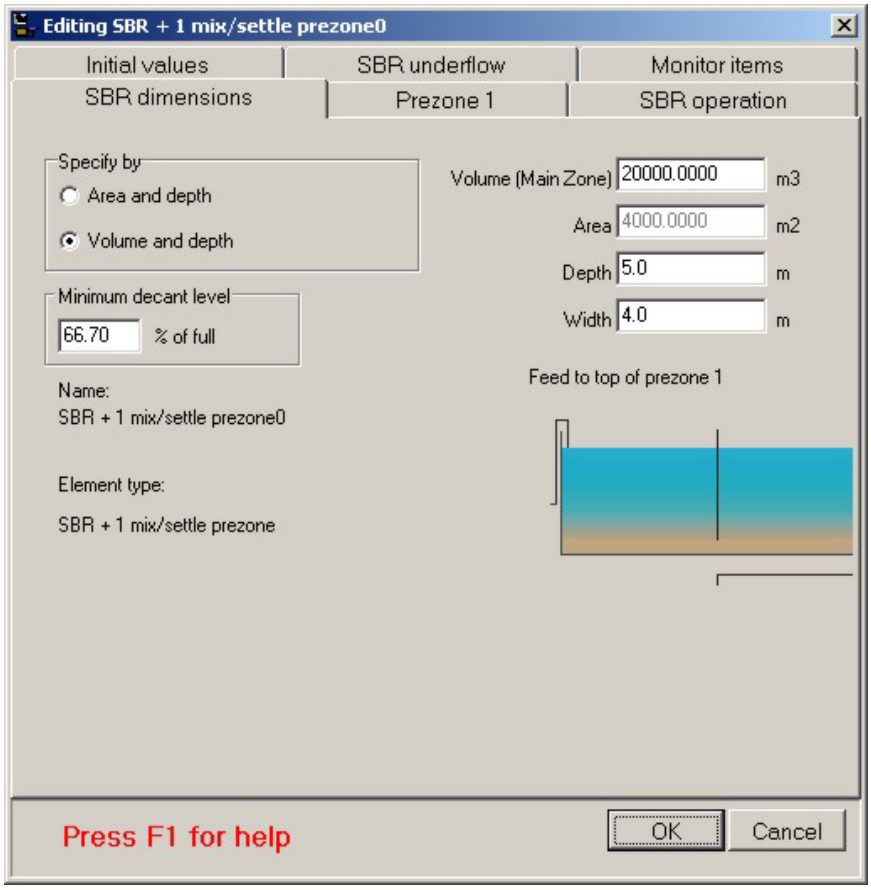
This type of SBR provides one hydraulically linked, mix/settle zone separated from the fill/draw SBR zone by a baffle. You may specify parameters related to the SBR dimensions, cycle operation, starting values, and underflow rates. You also may specify operating conditions for the mix/settle prezone. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The SBR with one mix/settle prezone monitor items tab

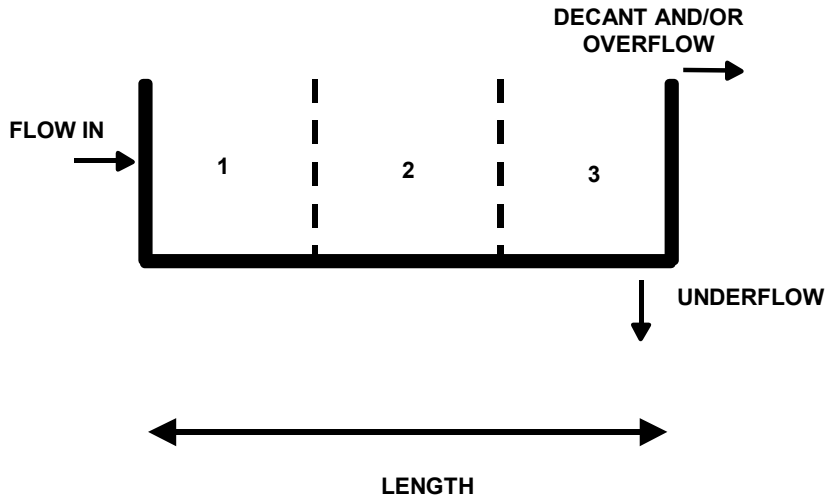
SBR + 1 Mix/Settle Prezone Dimensions

The **SBR Dimensions** tab, shown below, allows the user to enter the physical dimensions of the SBR zone of an SBR + 1 mix/settle prezone element.



The SBR zone dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the SBR zone area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the SBR zone volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Units are shown to the right of the edit boxes. Regardless of the method you choose, you also must specify a **Width** for the SBR zone.



Flow distribution in the SBR zone

BioWin uses the **Width** parameter to calculate the length of the SBR zone. The length is then divided into three equally sized zones. Underflow leaves the SBR zone at the bottom of the third zone, and decant and/or overflow leaves the SBR zone at the top of the third zone.

Some further explanation of how BioWin uses the SBR dimensions is warranted. When you set the dimensions, you specify a **Volume**, **Depth** and **Width** (and **Length** is calculated). Now think of a 2-d side-on view of the SBR with flow from left to right, as shown above.

For example, imagine the full depth is 5 m and the length is 15 m. When settling starts, this is what happens:

- The horizontal length of the SBR is divided into three equal-length subsections (each 5 m in this case).
- The vertical direction is divided into 10 equal-depth layers. [**Note:** as the level goes down during decanting - with no feed - the number of layers stays at 10, and the depth of each layer decreases]. That is, think of each length section as being 10 layers stacked on top of each other (and there are 3 sections).
- Each of the three sections is treated as a vertical settler. That is, the SBR (during settling) is 3 side-by-side settlers, with a total of 30 cells (each completely mixed i.e. of uniform composition).

Consider the dimensions of each of the 30 cells:

- The horizontal area of each cell is **Width * (Length/3) = W * 5 m** here.
- The end area of each cell is **Width * (Liquid Depth/10) = W * 0.5 m** (0.5 m when it is full). [**Note:** in this case the ratio of cell horizontal to

side area is 5 : 0.5 = 10 : 1 (if tank is full - but ratio increases when tank level drops)].

For the moment. assume that there is no inflow during decanting. Decant is removed from the top-right cell. During decanting several things happen (apart from settling in each of the 3 sections):

- Say 1 (incremental) unit of volume is removed from the top-right cell (at the composition of the top right cell).
- Overall, the volume of each of the 30 cells must be decreased by 1 unit / 30 (the length of each cell is fixed, so the depth must decrease).
- 29/30ths of the unit volume must get into the top right cell. That is, overall 1/30th of the volume must be removed from each of the other 29 cells.
- The 29/30ths gets into the top right cell from the cell to the left and from the cell below it.
- The cell at the bottom left of the SBR is a special case - there is no flow into it - only outflow of 1/30th. This flow leaves via the top surface and via the right hand end surface. The flow leaving each of the 2 surfaces is proportional to the areas of each surface.
- Flow balances around each of the other 28 cells is handled in a similar manner, doing a simultaneous flow balance on each cell. For the typical cell, there can be:
    - Flow out of the top surface;
    - Flow out the right side;
    - Flow in through the bottom surface; and
    - Flow in through the left surface.

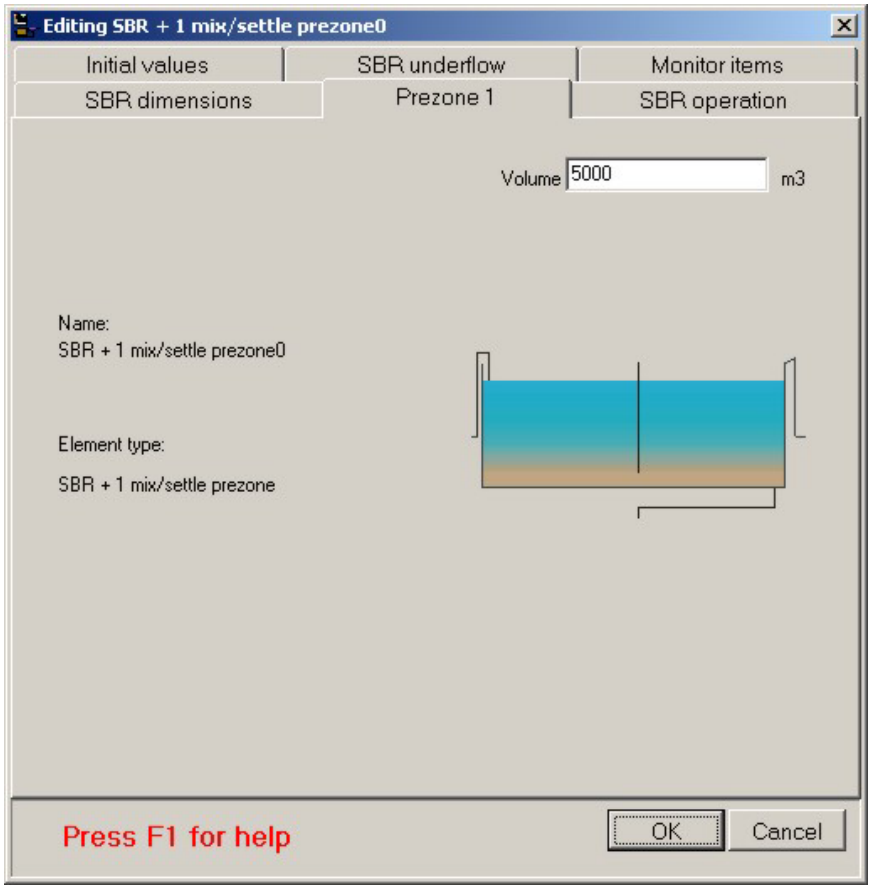
If there is influent flow during decanting, the flow balance is just slightly more complex, but happens in a similar fashion. Note that this explanation makes the process sound like a step-by-step calculation. In reality, what happens is a whole series of simultaneous integrations with error checking.

A **Minimum decant level** must also be specified as a percentage of the height. This level is the lowest liquid level that can be achieved by decanting liquid out of the top of the SBR zone. Note that it *is* possible to obtain lower liquid levels in a cycle than this value, but to get lower than this level, the liquid would have to go out via the underflow.

Note that unlike the SBR + 1 always-mixed prezone and SBR + 2 always-mixed prezones elements, you do not specify a feed layer – the feed is input to the top of a prezone and flow leaves from the bottom of a prezone.

Single Mix/Settle Prezone Setup

The **Prezone 1** tab shown below is used for specifying the volume of the prezone for the SBR + 1 mix/settle prezone element.

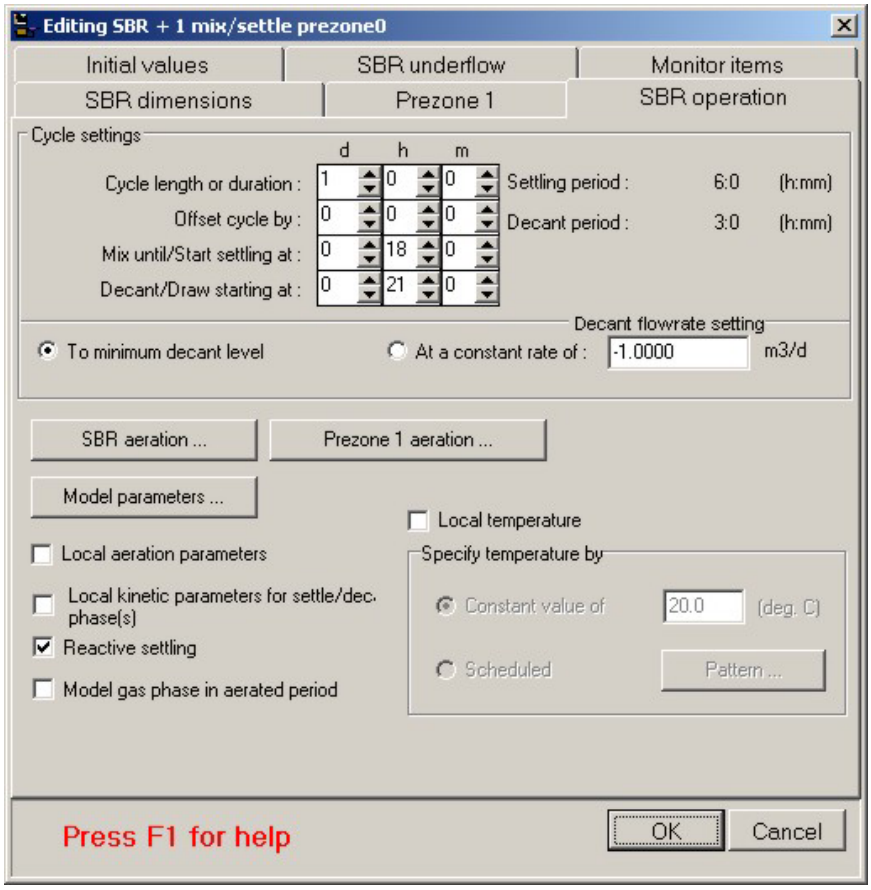


The single mix/settle prezone configuration tab

You can specify the volume of the prezone using the **Volume** text edit field.

SBR + 1 Mix/Settle Prezone SBR Zone Operation

The **SBR Operation** tab, shown below, allows the user to enter operating parameters for the SBR zone of an SBR + 1 mix/settle prezone element, such as cycle patterns, decant flow rates, aeration specifications and local temperature, as well as change local model parameters.



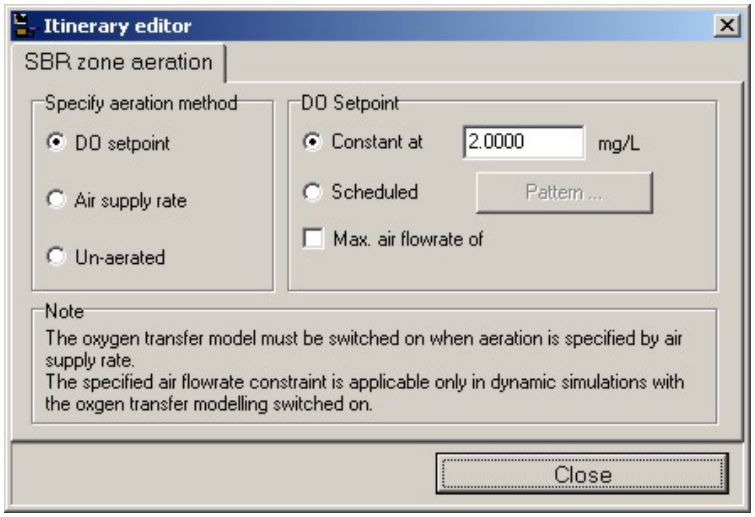
The SBR zone operation tab

The major operational settings of the SBR zone of the SBR + 1 mix/settle prezone are those related to **Cycle settings**. The **Cycle length or duration** is the total time that will elapse before the cycle begins to repeat itself. The first event to take place in the cycle is the end of the mixing phase/beginning of the settling phase. You specify this event time with the **Mix until/Start settling at:** spin edit control. Note that the dialog provides you with feedback as to the length of your settling period The second event to take place is the beginning of the decant phase. You specify this event time with the **Decant/Draw starting at:** spin edit control. Note that the dialog provides you with feedback as to the length of your decant period. Experimenting with these settings will reveal that settling and decanting take place simultaneously – but the settling phase length always must be greater than the decant phase length. Finally, you may set up a cycle offset if you wish using the **Offset cycle by:** control.

Note that BioWin assumes that there is no influent or underflow activity during the decant period and bases the calculation for required decant flowrate on that premise.

The next operational setting is the **Decant flowrate setting**. You may specify this with one of two possible methods. If you know the flowrate you wish to decant at, then choose the **At a constant rate of:** option and enter the desired value. If you wish BioWin to calculate the decant flowrate for you, select the **To minimum decant level** option. The decant flowrate is calculated as follows. At the beginning of the decant period, the simulator looks at the difference between the current liquid volume and the specified minimum decant volume. It then calculates the necessary flowrate to decant the available liquid during the decant period.

You can specify the SBR zone aeration settings by clicking the **SBR aeration…** button which will show the following dialog box.

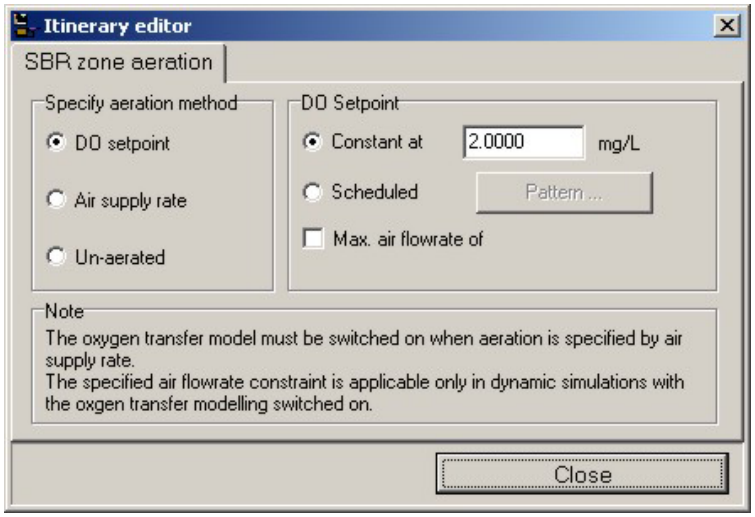


The SBR zone aeration setting dialog box

There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **SBR DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **SBR air flow itinerary** dialog box).

You can set up the prezone aeration operation by clicking the **Prezone 1 aeration…** button. Doing so will present you with the dialog box shown below.



The single mix/settle prezone aeration settings dialog box

There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be

specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **SBR DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **SBR air flow itinerary** dialog box).

Note that if you wish to change any of the aeration model parameters, click the **Model parameters…** button in the main **Operation** tab to open the **Model parameter editor** dialog box (Note that this changes the aeration model parameters on a local level only).

A **Local temperature** also may be specified for an SBR + 1 mix/settle prezone element. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

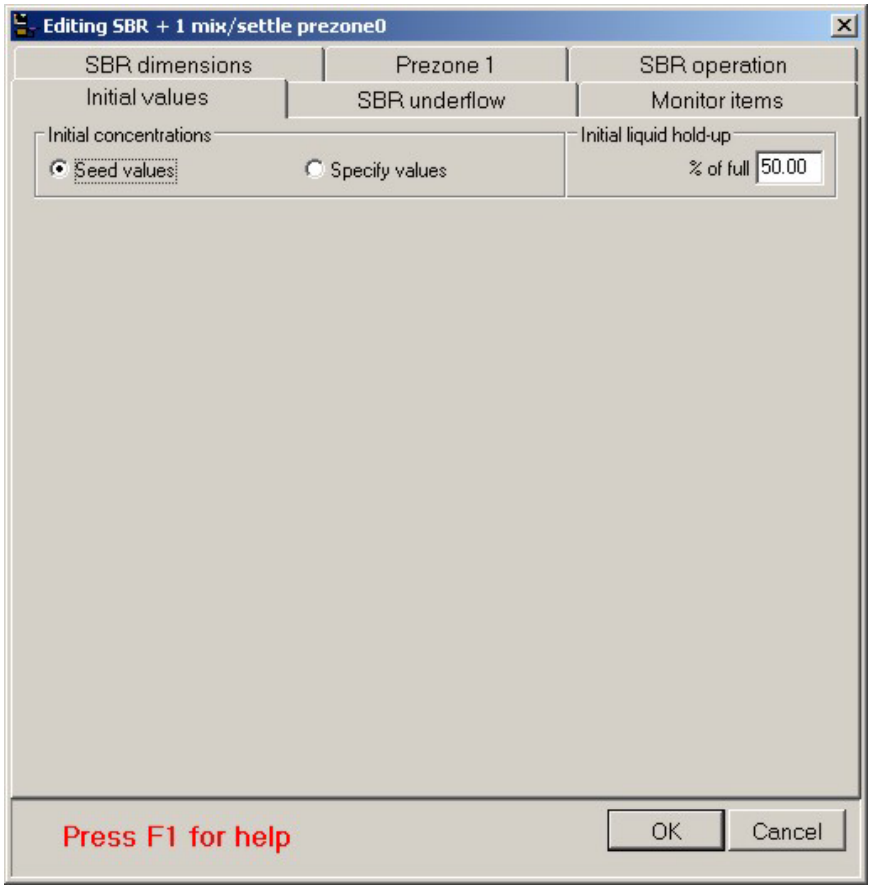
You also can specify **Local kinetic parameters** for the activated sludge model used in the prezone and SBR zone **decant/settling phase**. For the mixing phase, the global model parameter values are used. If you click on the check box for local kinetic parameters, then clicking the **Model parameters…** button opens the **Model parameter editor** dialog box allowing you access to the various activated sludge model kinetic parameters.

To specify local aeration parameters (e.g. alpha and beta factors), check the box labeled **Local aeration parameters**.

To model the concentrations of the constituents of the aeration gas (e.g. oxygen, carbon dioxide, etc.), check the box labeled **Model gas phase**. If this option is selected you may also enter a percentage value for the **Gas hold-up** (i.e. the percentage of the total reactor volume occupied by aeration gas).

SBR + 1 Mix/Settle Prezone Initial Values

The **Initial values** tab, shown below, is used for setting up the initial settings for SBR + 1 mix/settle prezone concentrations and volume.

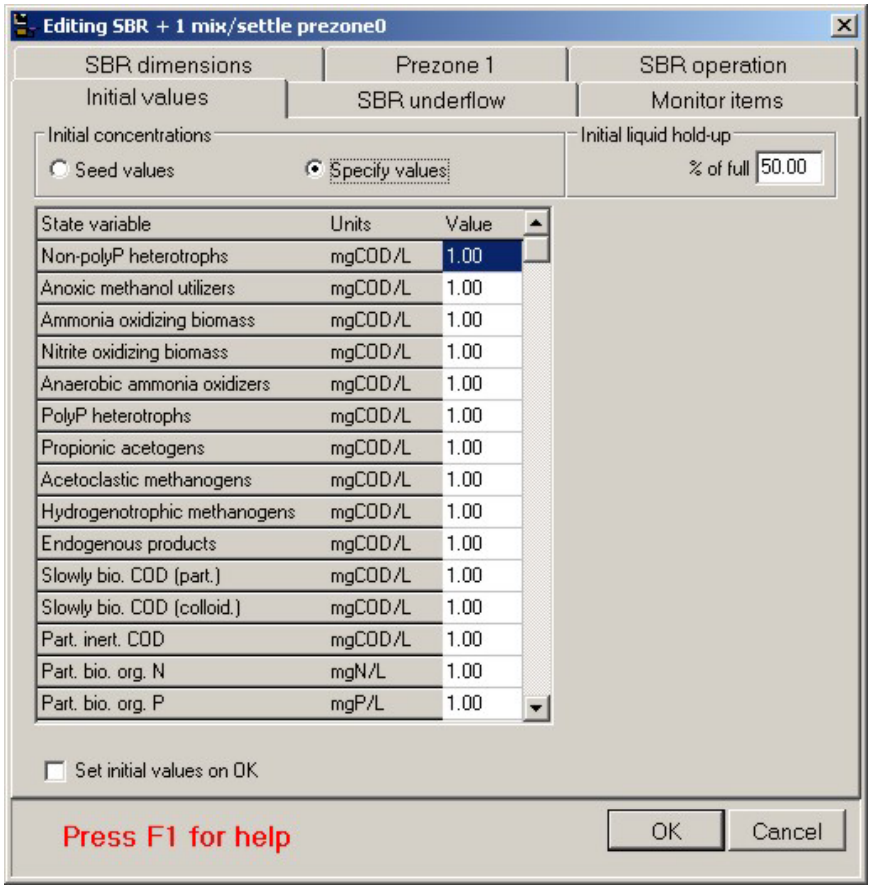


The SBR + 1 mix/settle prezone initial values tab with the Seed values *option selected*

Use the **Initial liquid hold-up** field to enter a value to specify the **% of full** setting. It should be noted here that the lower and upper limits on this value are 0.02 and 99.98%, respectively. This reflects the fact that this is the liquid volume, and allows for small differences between *liquid* volume and *reactor* volume for inlet and outlet pipes.

There are two methods for setting up **Initial concentrations** in the SBR + 1 mix/settle prezone. The dialog box shown above illustrates the first case where the **Seed values** option is selected. When this option is selected, BioWin applies default seed values for all the state variables (except volume, which you specify) in the same manner it would for other bioreactor elements *in all of the zones* (i.e. including the prezone).

The dialog box shown below illustrates the second case where the **Specify values** option is selected. In this case, an editable list of state variables is displayed in the dialog box. This allows you to enter your own seed values for state variables in the SBR + 1 mix/settle prezone. These will be inserted in each zone of the SBR + 1 mix/settle prezone element when you begin a dynamic simulation and choose to use seed values.



The SBR + 1 mix/settle prezone initial values tab with the Specify values *option selected*

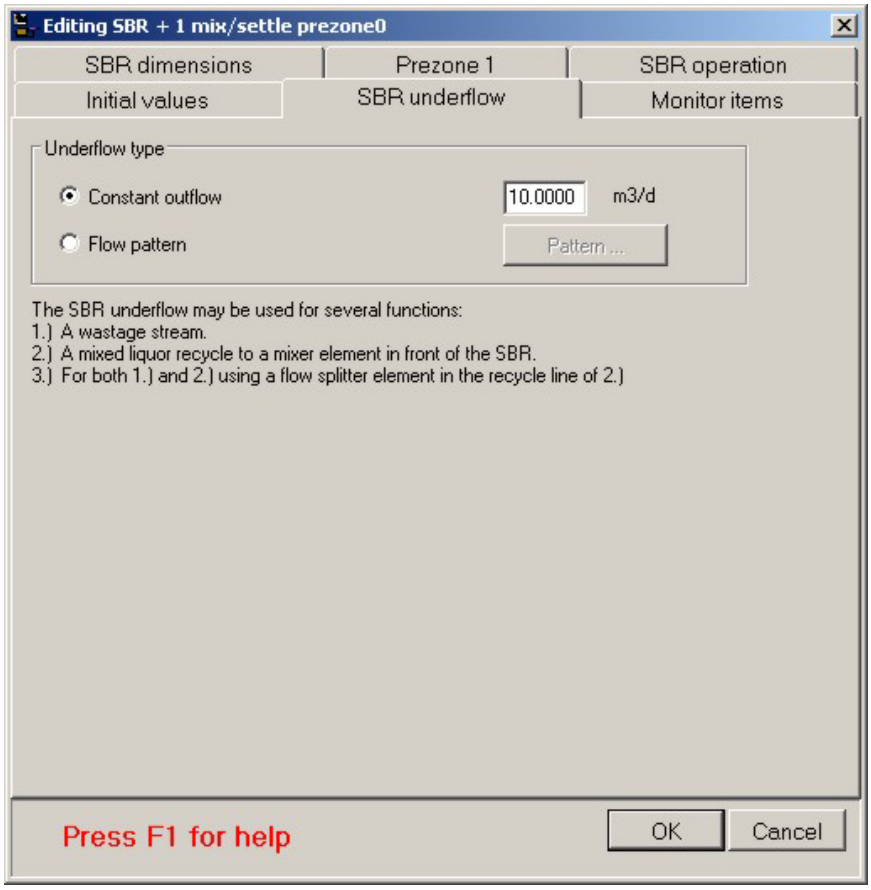
When the specified seed values get inserted in the SBR + 1 mix/settle prezone is determined by the **Set initial values on OK** option. If this option **is not** checked:

- The specified values will be placed in the SBR + 1 mix/settle prezone at the beginning of a dynamic simulation **if** you elect to seed the simulation.

If this option **is** checked, the specified values are placed in the SBR + 1 mix/settle prezone when you click **OK** to close the dialog box, **overriding any existing state variable values**. Therefore, the state variable values at the beginning of a dynamic simulation will be the same regardless of whether or not you choose to re-seed.

SBR + 1 Mix/Settle Prezone Underflow

The SBR + 1 mix/settle prezone **SBR Underflow** tab shown below allows you to set up underflow rates and patterns for the SBR zone of this element.



The SBR zone underflow settings tab

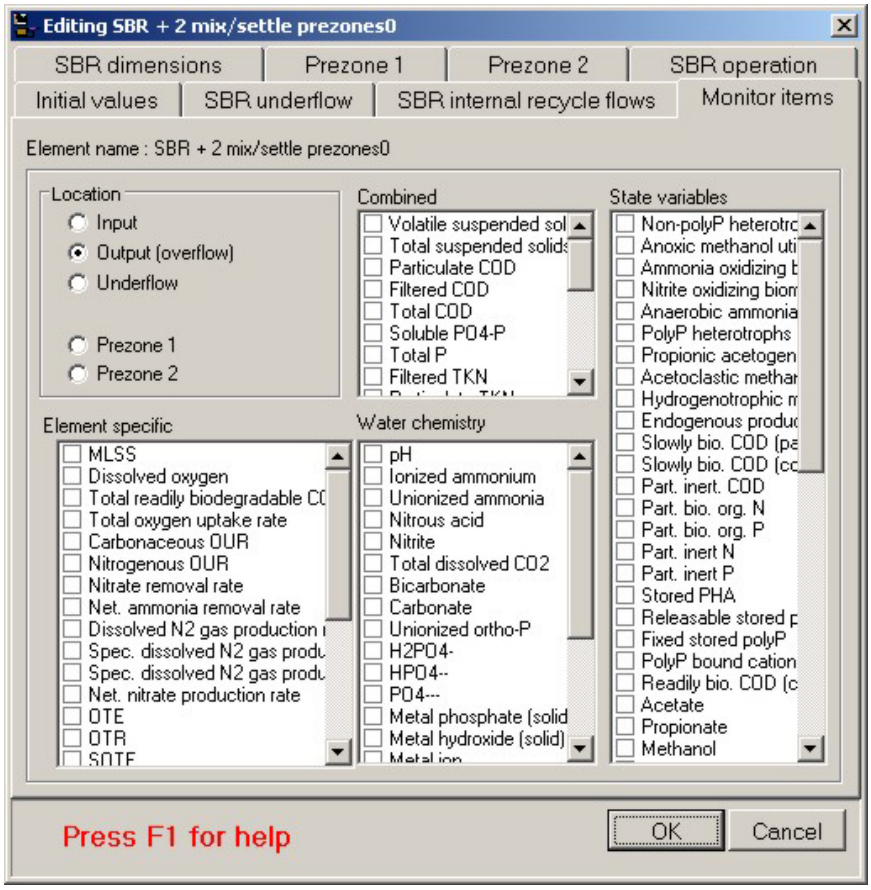
The **Underflow type** may be specified as a **Constant underflow** or a **Flow pattern** by selecting the appropriate option. If you select **Constant underflow**, you may enter the desired underflow rate in the edit field. If you select **Flow pattern**, clicking the **Pattern…** button will open the **Underflow rate itinerary** dialog box.

The SBR zone underflow may be used for several functions:

1. As a wastage stream;
2. A mixed liquor recycle to a mixer element in front of the SBR + 1 mix/settle prezone element;
3. As both a wastage and a recycle by placing a splitter element between the underflow and the mixer element in front of the SBR + 1 mix/settle prezone element.

SBR with 2 Mix/Settle Prezones

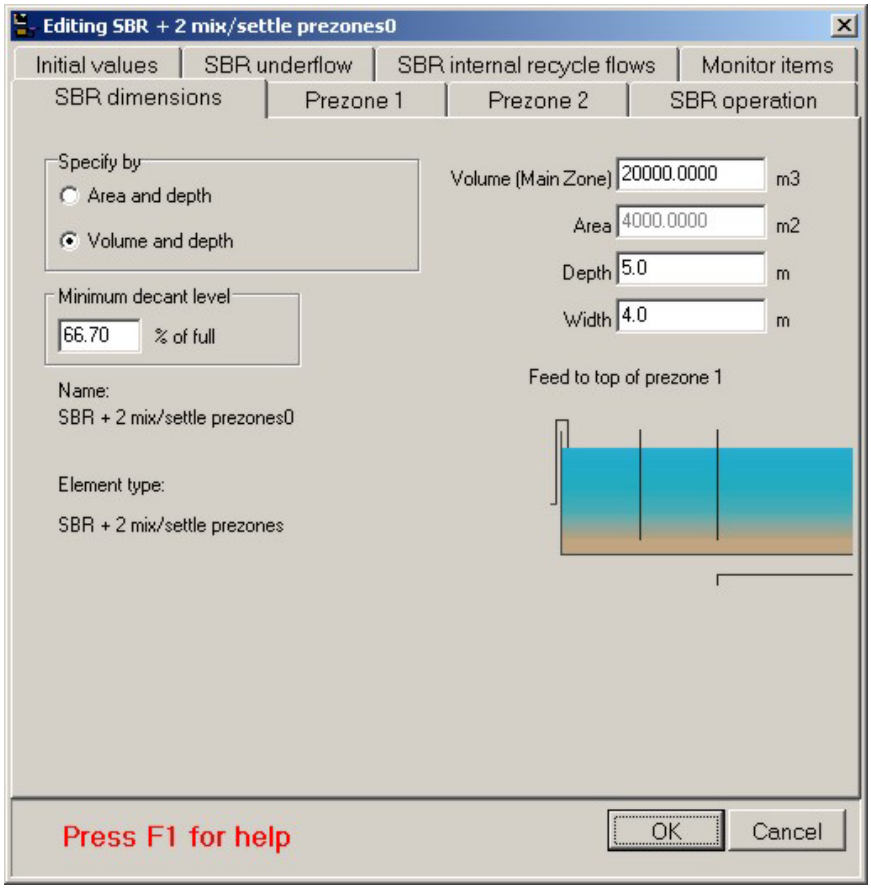
This type of SBR provides two hydraulically linked, mix/settle zones separated from the fill/draw SBR zone by a baffle. You may specify parameters related to the SBR dimensions, cycle operation, starting values, and underflow rates. You also may specify operating conditions for the mix/settle prezones. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The SBR with two mix/settle prezones monitor items tab

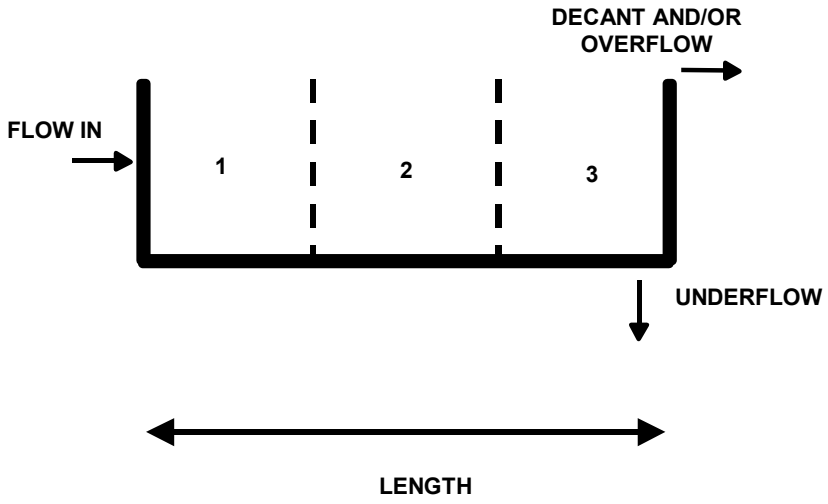
SBR + 2 Mix/Settle Prezones Dimensions

The **SBR Dimensions** tab, shown below, allows the user to enter the physical dimensions of the SBR zone of an SBR + 2 mix/settle prezones element.



The SBR zone dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the SBR zone area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the SBR zone volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Units are shown to the right of the edit boxes. Regardless of the method you choose, you also must specify a **Width** for the SBR zone.



Flow distribution in the SBR zone

BioWin uses the **Width** parameter to calculate the length of the SBR zone. The length is then divided into three equally sized zones. Underflow leaves the SBR zone at the bottom of the third zone, and decant and/or overflow leaves the SBR zone at the top of the third zone.

Some further explanation of how BioWin uses the SBR dimensions is warranted. When you set the dimensions, you specify a **Volume**, **Depth** and **Width** (and **Length** is calculated). Now think of a 2-d side-on view of the SBR with flow from left to right, as shown above.

For example, imagine the full depth is 5 m and the length is 15 m. When settling starts, this is what happens:

- The horizontal length of the SBR is divided into three equal-length subsections (each 5 m in this case).
- The vertical direction is divided into 10 equal-depth layers. [**Note:** as the level goes down during decanting - with no feed - the number of layers stays at 10, and the depth of each layer decreases]. That is, think of each length section as being 10 layers stacked on top of each other (and there are 3 sections).
- Each of the three sections is treated as a vertical settler. That is, the SBR (during settling) is 3 side-by-side settlers, with a total of 30 cells (each completely mixed i.e. of uniform composition).

Consider the dimensions of each of the 30 cells:

- The horizontal area of each cell is **Width * (Length/3) = W * 5 m** here.
- The end area of each cell is **Width * (Liquid Depth/10) = W * 0.5 m** (0.5 m when it is full). [**Note:** in this case the ratio of cell horizontal to

side area is 5 : 0.5 = 10 : 1 (if tank is full - but ratio increases when tank level drops)].

For the moment. assume that there is no inflow during decanting. Decant is removed from the top-right cell. During decanting several things happen (apart from settling in each of the 3 sections):

- Say 1 (incremental) unit of volume is removed from the top-right cell (at the composition of the top right cell).
- Overall, the volume of each of the 30 cells must be decreased by 1 unit / 30 (the length of each cell is fixed, so the depth must decrease).
- 29/30ths of the unit volume must get into the top right cell. That is, overall 1/30th of the volume must be removed from each of the other 29 cells.
- The 29/30ths gets into the top right cell from the cell to the left and from the cell below it.
- The cell at the bottom left of the SBR is a special case - there is no flow into it - only outflow of 1/30th. This flow leaves via the top surface and via the right hand end surface. The flow leaving each of the 2 surfaces is proportional to the areas of each surface.
- Flow balances around each of the other 28 cells is handled in a similar manner, doing a simultaneous flow balance on each cell. For the typical cell, there can be:
    - Flow out of the top surface;
    - Flow out the right side;
    - Flow in through the bottom surface; and
    - Flow in through the left surface.

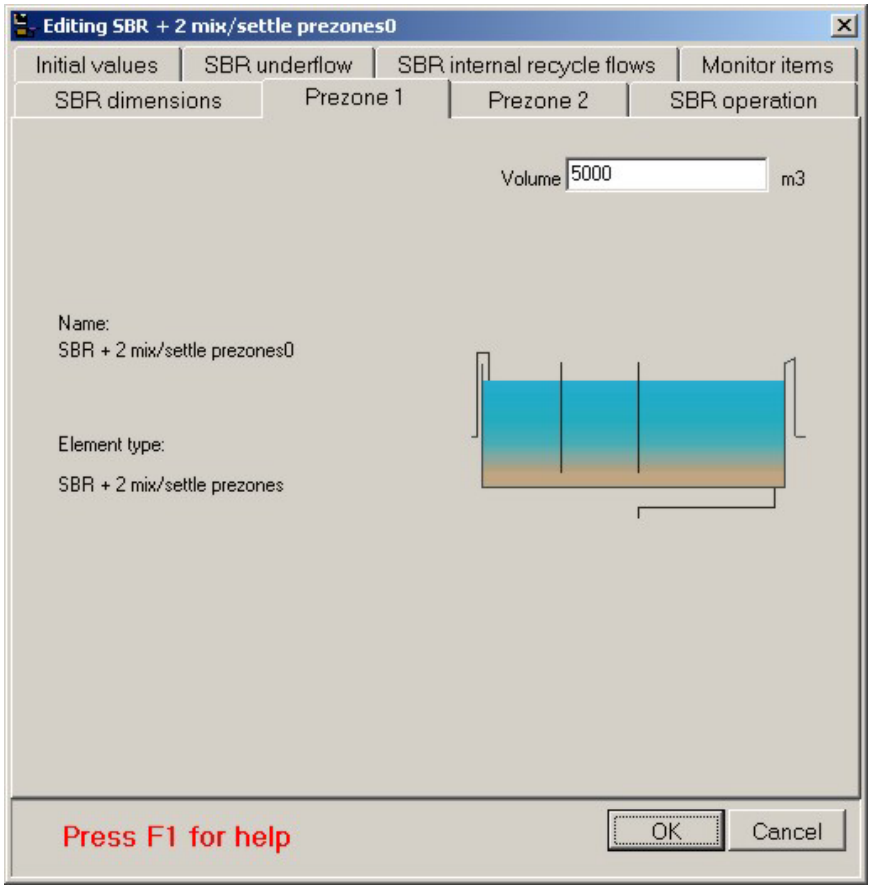
If there is influent flow during decanting, the flow balance is just slightly more complex, but happens in a similar fashion. Note that this explanation makes the process sound like a step-by-step calculation. In reality, what happens is a whole series of simultaneous integrations with error checking.

A **Minimum decant level** must also be specified as a percentage of the height. This level is the lowest liquid level that can be achieved by decanting liquid out of the top of the SBR zone. Note that it *is* possible to obtain lower liquid levels in a cycle than this value, but to get lower than this level, the liquid would have to go out via the underflow.

Note that unlike the SBR + 1 always-mixed prezone and SBR + 2 always-mixed prezones elements, you *do not* specify a feed layer – the feed is input at the top of the first prezone.

First Mix/Settle Prezone Setup

The **Prezone 1** tab shown below is used for specifying the volume of the first prezone for the SBR + 2 mix/settle prezones element.

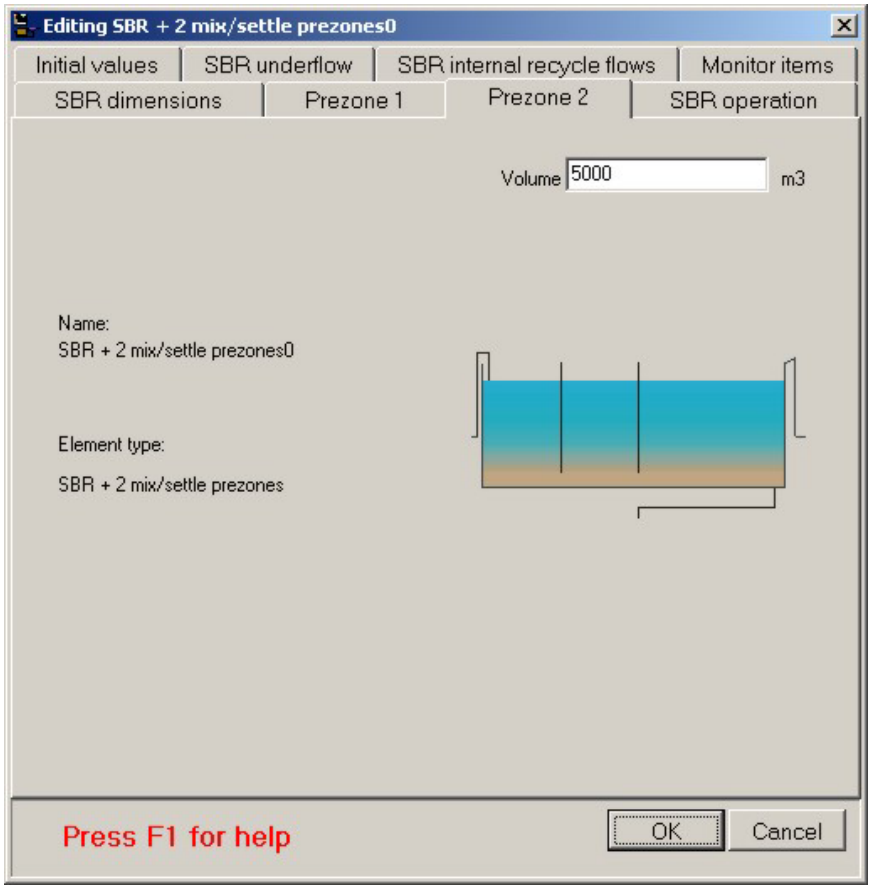


The first mix/settle prezone configuration tab

You can specify the volume of the prezone using the **Volume** text edit field.

Second Mix/Settle Prezone Setup

The **Prezone 2** tab shown below is used for specifying the volume of the second prezone for the SBR + 2 mix/settle prezones element.

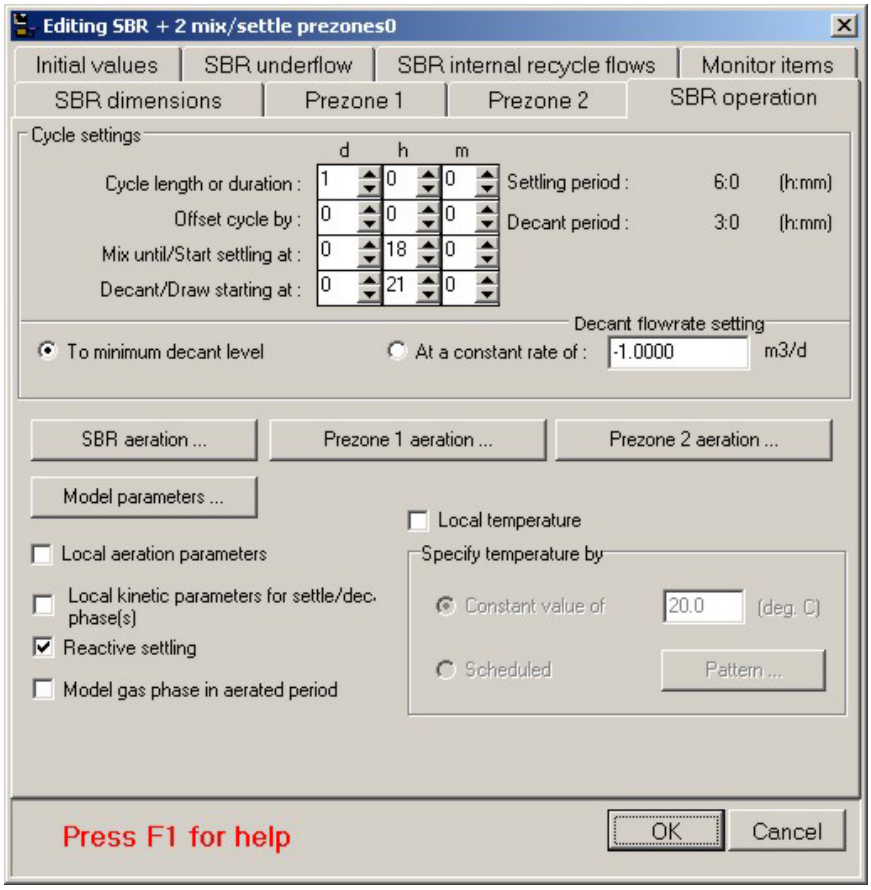


The second mix/settle prezone configuration tab

You can specify the volume of the prezone using the **Volume** text edit field.

SBR + 2 Mix/Settle Prezones SBR Zone Operation

The **SBR Operation** tab, shown below, allows the user to enter operating parameters for the SBR zone of an SBR + 2 mix/settle prezones element, such as cycle patterns, decant flow rates, aeration specifications and local temperature, as well as change local model parameters.



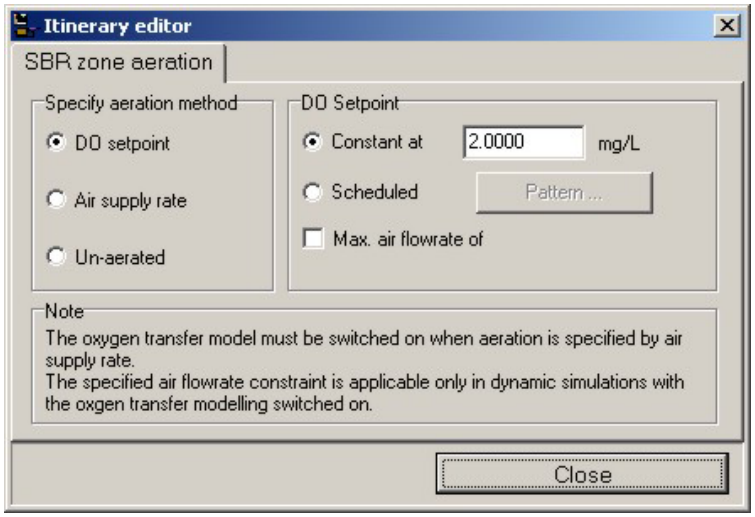
The SBR zone operation tab

The major operational settings of the SBR zone of the SBR + 2 mix/settle prezones are those related to **Cycle settings**. The **Cycle length or duration** is the total time that will elapse before the cycle begins to repeat itself. The first event to take place in the cycle is the end of the mixing phase/beginning of the settling phase. You specify this event time with the **Mix until/Start settling at:** spin edit control. Note that the dialog provides you with feedback as to the length of your settling period The second event to take place is the beginning of the decant phase. You specify this event time with the **Decant/Draw starting at:** spin edit control. Note that the dialog provides you with feedback as to the length of your decant period. Experimenting with these settings will reveal that settling and decanting take place simultaneously – but the settling phase length always must be greater than the decant phase length. Finally, you may set up a cycle offset if you wish using the **Offset cycle by:** control.

Note that BioWin assumes that there is no influent or underflow activity during the decant period and bases the calculation for required decant flowrate on that premise.

The next operational setting is the **Decant flowrate setting**. You may specify this with one of two possible methods. If you know the flowrate you wish to decant at, then choose the **At a constant rate of:** option and enter the desired value. If you wish BioWin to calculate the decant flowrate for you, select the **To minimum decant level** option. The decant flowrate is calculated as follows. At the beginning of the decant period, the simulator looks at the difference between the current liquid volume and the specified minimum decant volume. It then calculates the necessary flowrate to decant the available liquid during the decant period.

You can specify the SBR zone aeration settings by clicking the **SBR aeration…** button which will show the following dialog box.

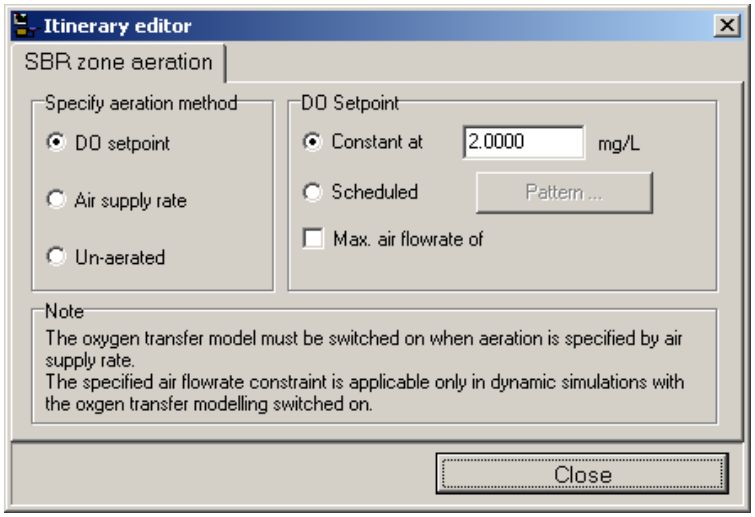


The SBR zone aeration setting dialog box

There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **SBR DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **SBR air flow itinerary** dialog box).

You can set up the first prezone aeration operation by clicking the **Prezone 1 aeration…** button. Doing so will present you with the dialog box shown below.



The first mix/settle prezone aeration settings dialog box

There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be

specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **SBR DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **SBR air flow itinerary** dialog box).

Clicking the **Prezone 2 aeration…** button allows you to specify aeration settings for the second prezone using the same dialog boxes as those used for the first zone, shown above.

Note that if you wish to change any of the aeration model parameters, click the **Model parameters…** button in the main **Operation** tab to open the **Model parameter editor** dialog box (Note that this changes the aeration model parameters on a local level only).

A **Local temperature** also may be specified for an SBR + 2 mix/settle prezones element. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

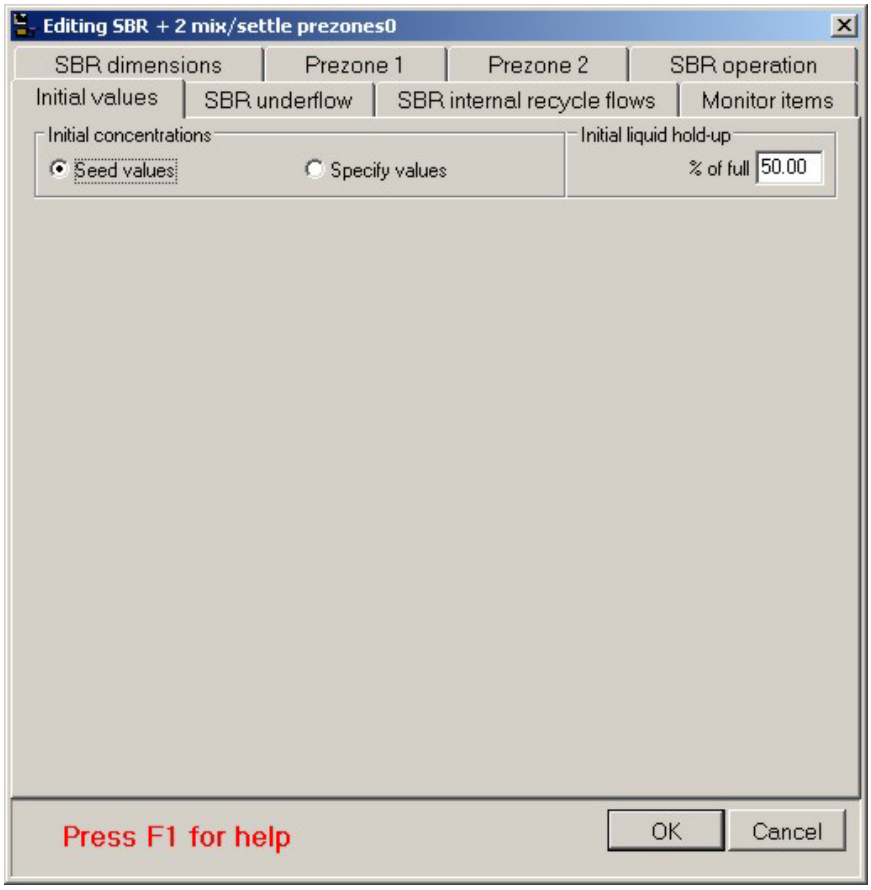
You also can specify **Local kinetic parameters** for the activated sludge model used in the prezones and SBR zone **decant/settling phase**. For the mixing phase, the global model parameter values are used. If you click on the check box for local kinetic parameters, then clicking the **Model parameters…** button opens the **Model parameter editor** dialog box allowing you access to the various activated sludge model kinetic parameters.

To specify local aeration parameters (e.g. alpha and beta factors), check the box labeled **Local aeration parameters**.

To model the concentrations of the constituents of the aeration gas (e.g. oxygen, carbon dioxide, etc.), check the box labeled **Model gas phase**. If this option is selected you may also enter a percentage value for the **Gas hold-up** (i.e. the percentage of the total reactor volume occupied by aeration gas).

SBR + 2 Mix/Settle Prezones Initial Values

The **Initial values** tab, shown below, is used for setting up the initial settings for SBR + 2 mix/settle prezones concentrations and volume.

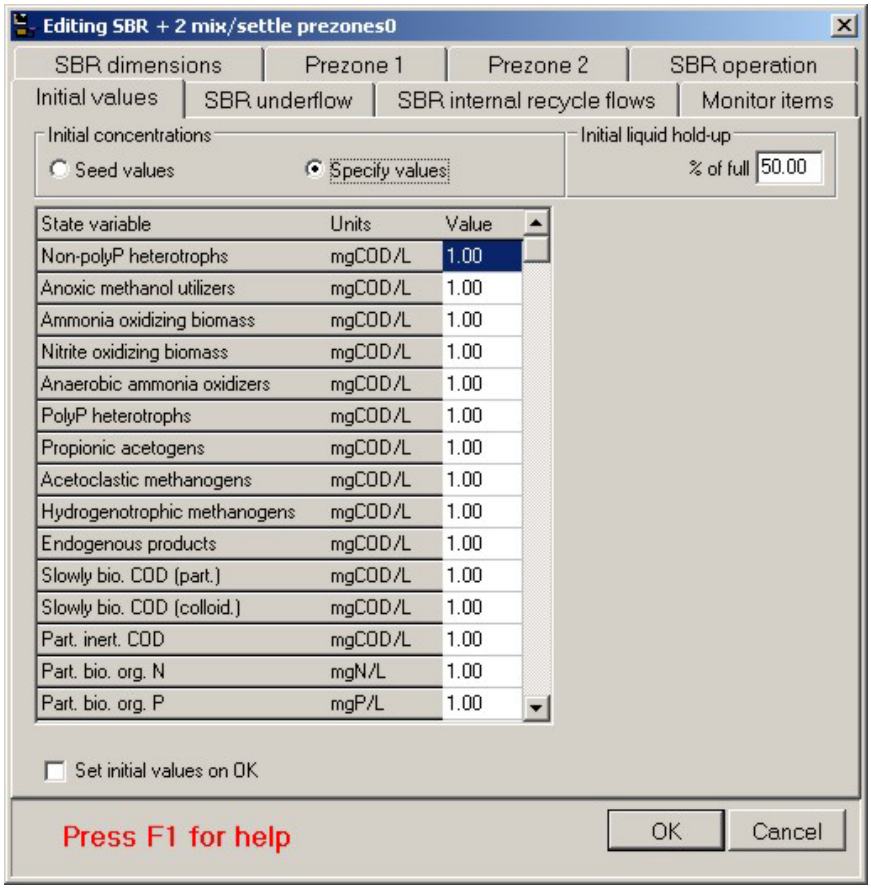


The SBR + 2 mix/settle prezones initial values tab with the Seed values *option selected*

Use the **Initial liquid hold-up** field to enter a value to specify the **% of full** setting. It should be noted here that the lower and upper limits on this value are 0.02 and 99.98%, respectively. This reflects the fact that this is the liquid volume, and allows for small differences between *liquid* volume and *reactor* volume for inlet and outlet pipes.

There are two methods for setting up **Initial concentrations** in the SBR + 2 mix/settle prezones. The dialog box shown above illustrates the first case where the **Seed values** option is selected. When this option is selected, BioWin applies default seed values for all the state variables (except volume, which you specify) in the same manner it would for other bioreactor elements *in all of the zones* (i.e. including the prezones).

The dialog box shown below illustrates the second case where the **Specify values** option is selected. In this case, an editable list of state variables is displayed in the dialog box. This allows you to enter your own seed values for state variables in the SBR + 2 mix/settle prezones. These will be inserted in each zone of the SBR + 2 mix/settle prezones element when you begin a dynamic simulation and choose to use seed values.



The SBR + 2 mix/settle prezones initial values tab with the Specify values *option selected*

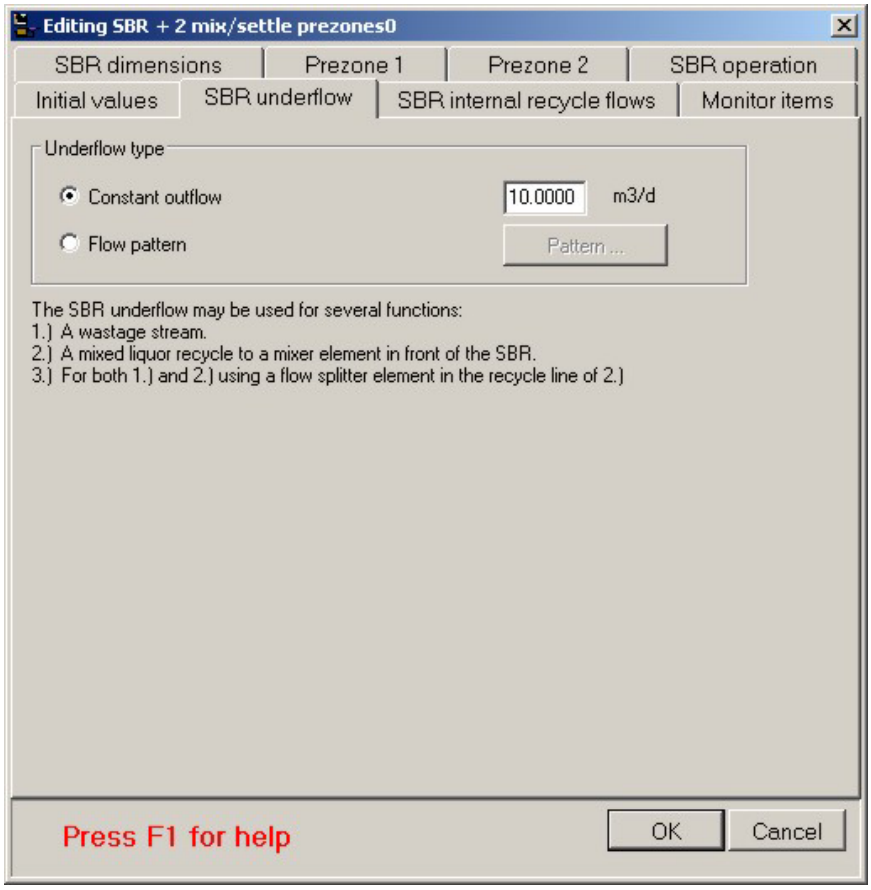
When the specified seed values get inserted in the SBR + 2 mix/settle prezones is determined by the **Set initial values on OK** option. If this option **is not** checked:

- The specified values will be placed in the SBR + 2 mix/settle prezones at the beginning of a dynamic simulation **if** you elect to seed the simulation.

If this option **is** checked, the specified values are placed in the SBR + 2 mix/settle prezones when you click **OK** to close the dialog box, **overriding any existing state variable values**. Therefore, the state variable values at the beginning of a dynamic simulation will be the same regardless of whether or not you choose to re-seed.

SBR + 2 Mix/Settle Prezones Underflow

The SBR + 2 mix/settle prezones **SBR Underflow** tab shown below allows you to set up underflow rates and patterns for the SBR zone.



The SBR zone underflow settings tab

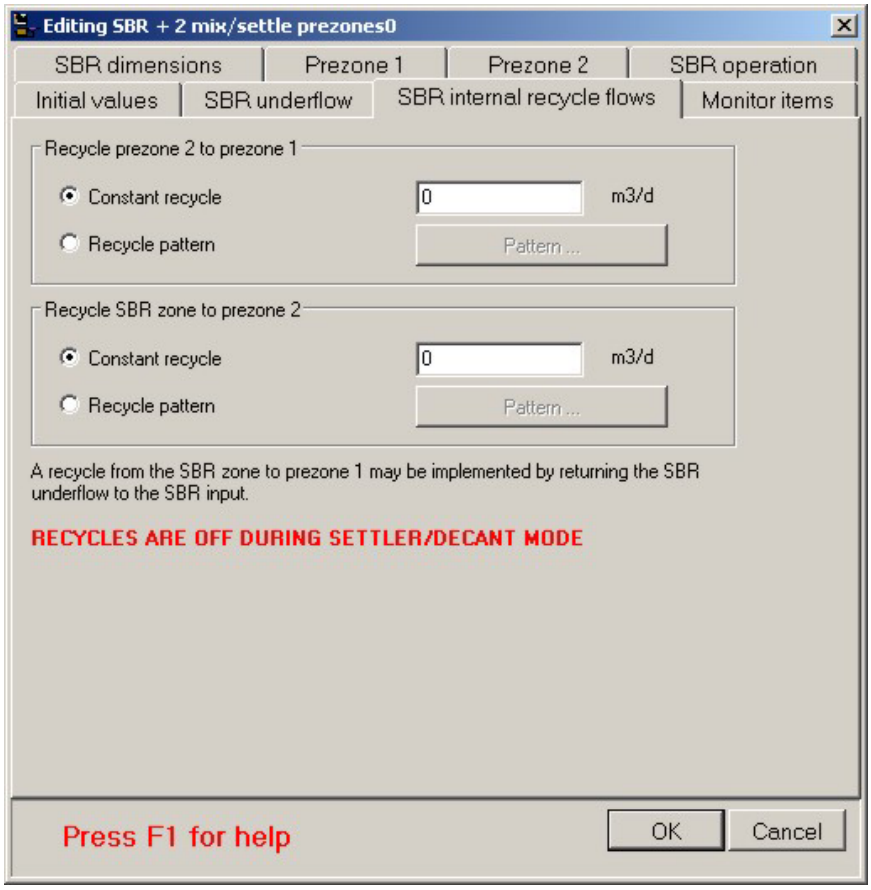
The **Underflow type** may be specified as a **Constant underflow** or a **Flow pattern** by selecting the appropriate option. If you select **Constant underflow**, you may enter the desired underflow rate in the edit field. If you select **Flow pattern**, clicking the **Pattern…** button will open the **Underflow rate itinerary** dialog box.

The SBR zone underflow may be used for several functions:

1. As a wastage stream;
2. A mixed liquor recycle to a mixer element in front of the SBR + 2 mix/settle prezones element;
3. As both a wastage and a recycle by placing a splitter element between the underflow and the mixer element in front of the SBR + 2 mix/settle prezones element.

SBR + 2 Mix/Settle Prezones Internal Recycle Flows

The SBR + 2 mix/settle prezones element **SBR internal recycle flows** tab shown below allows you to set up recycles between the SBR zone and the second prezone, and between the second and first prezone.



The SBR internal recycle tab

You may have a recycle flow which goes from the second prezone to the first one. If you select **Constant recycle**, you may enter the desired recycle flowrate in the edit field. If you select **Recycle pattern**, clicking the **Pattern…** button will open the **Recycle flowrate itinerary** dialog box.

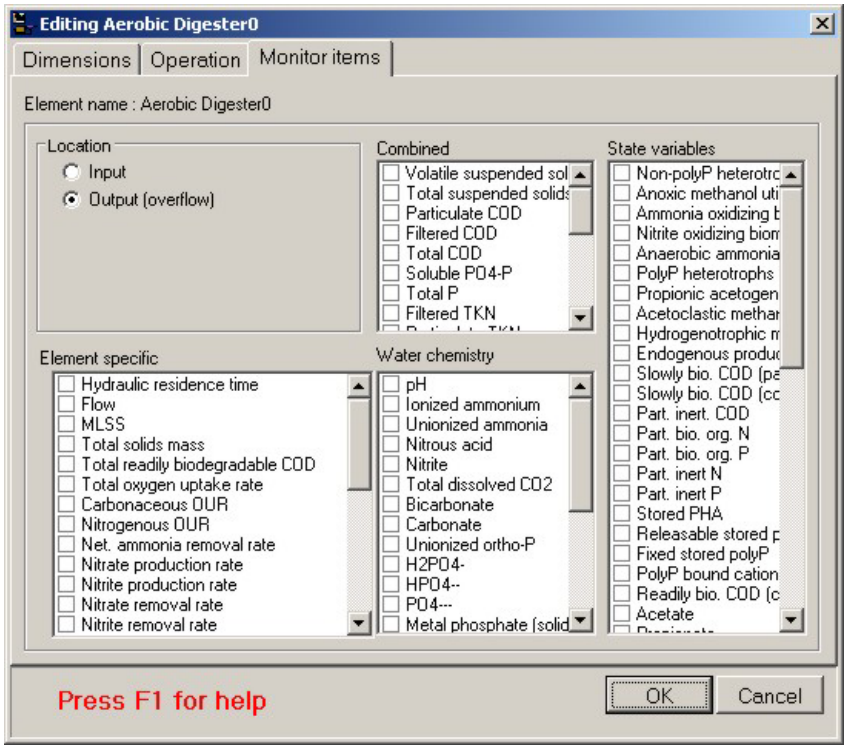
You also may have a recycle flow which goes from the SBR zone to the second prezone. If you select **Constant recycle**, you may enter the desired recycle flowrate in the edit field. If you select **Recycle pattern**, clicking the **Pattern…** button will open the **Recycle flowrate itinerary** dialog box.

Two final points should be noted with regard to internal recycle flows:

1. It is possible to have a recycle flow from the SBR zone to the first prezone by returning the SBR zone underflow (or a portion of it) to the input of the SBR + 2 mix/settle prezones element.
2. Internal recycle flows are set to zero when the SBR zone goes into settle/decant mode.

Aerobic Digester

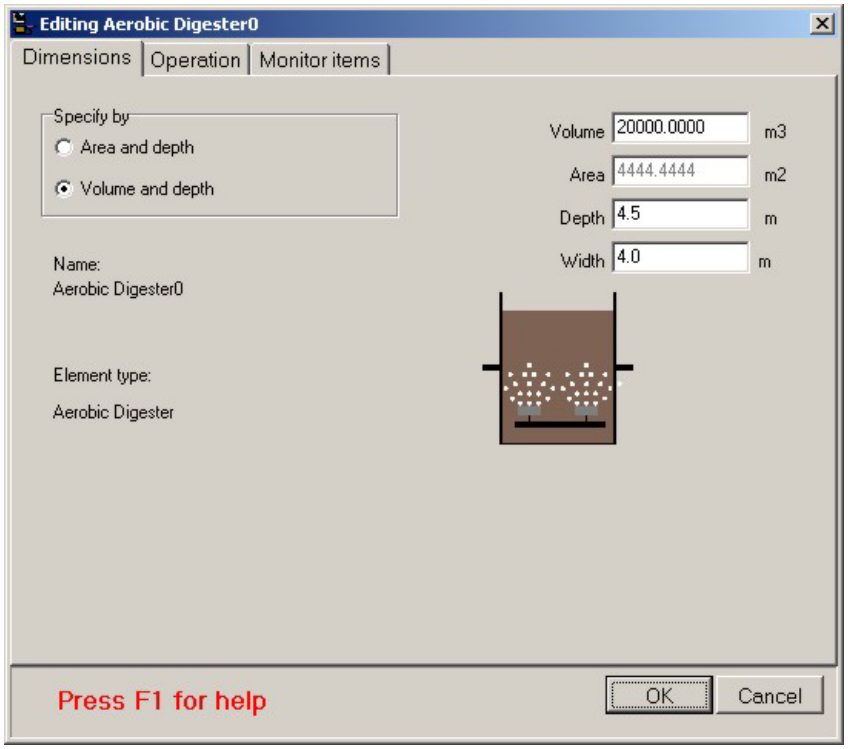
The aerobic digester simulates the aerobic digestion process in a CSTR. You may specify parameters related to the operation and control of the aerobic digester element. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The aerobic digester monitor items tab

Aerobic Digester Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of an aerobic digester element.

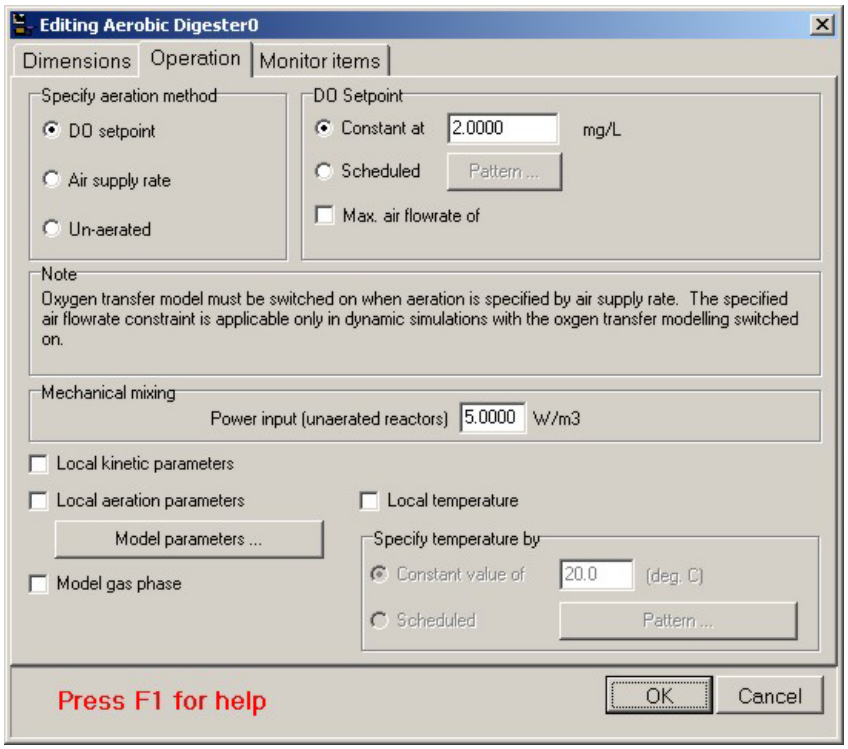


The aerobic digester dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the aerobic digester area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the aerobic digester volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a **Width** for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

Aerobic Digester Operation

The **Operation** tab, shown below, allows the user to enter operating parameters for an aerobic digester element, such as aeration specifications and local temperature, as well as change local model parameters.



The aerobic digester operation tab

There are two methods for specifying aeration: either by a **DO setpoint** or by an **Air supply rate**. The aeration method is specified by clicking on the appropriate radio button. When **DO setpoint** is selected, the setpoint concentration must be specified. You may specify either a **Constant** setpoint or a **Scheduled** DO setpoint pattern (clicking the **Pattern…** button will open the **Edit DO setpoint itinerary** dialog box). You may wish to place a restriction on the maximum allowable air flowrate that may be used to achieve the desired DO setpoint. This is a useful feature for investigating the ability of air equipment to achieve desired DO setpoints.

When **Air supply rate** is selected, the air flowrate must be specified. You may specify either a **Constant** air flowrate or a **Scheduled** air flowrate pattern (clicking the **Pattern…** button will open the **Edit air flow itinerary** dialog box).

The mechanical mixing **Power input (unaerated reactors)** can be specified. This is used to calculate the velocity gradient in the vessel.

Note : If you wish to change any of the aeration model parameters, click the **Model parameters…** button to open the **Model parameter editor** dialog box (Note that this changes the aeration model parameters on a local level only).

A **Local temperature** also may be specified for a aerobic digester element. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

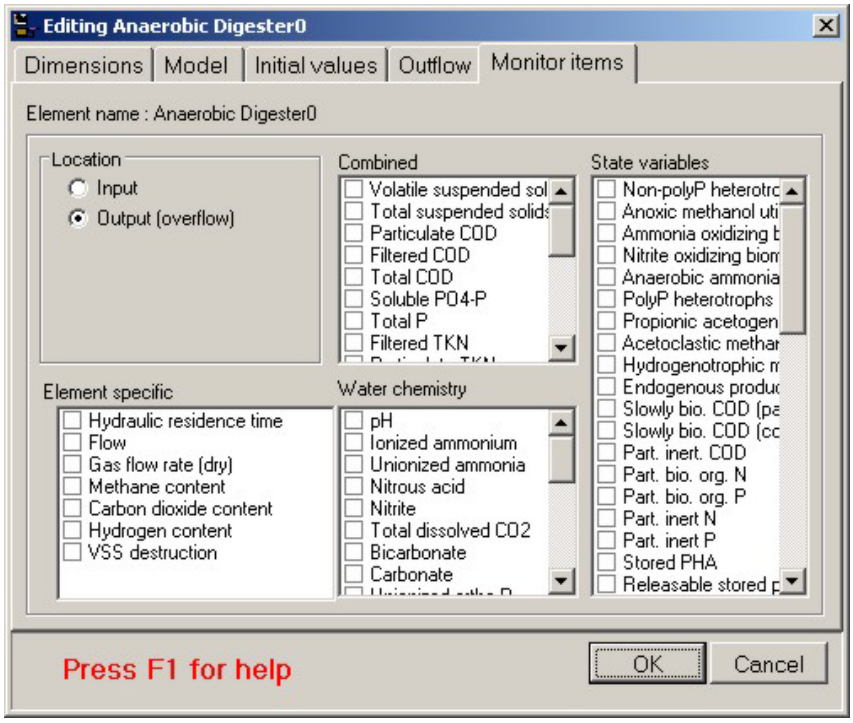
You also can specify **Local kinetic parameters** for the activated sludge model used in the aerobic digester element. If you click on the check box for local kinetic parameters, then clicking the **Model parameters…** button opens the **Model parameter editor** dialog box allowing you access to the various activated sludge model kinetic parameters.

To specify local aeration parameters (e.g. alpha and beta factors), check the box labeled **Local aeration parameters**.

To model the concentrations of the constituents of the aeration gas (e.g. oxygen, carbon dioxide, etc.), check the box labeled **Model gas phase**. If this option is selected you may also enter a percentage value for the **Gas hold-up** (i.e. the percentage of the total reactor volume occupied by aeration gas).

Anaerobic Digester

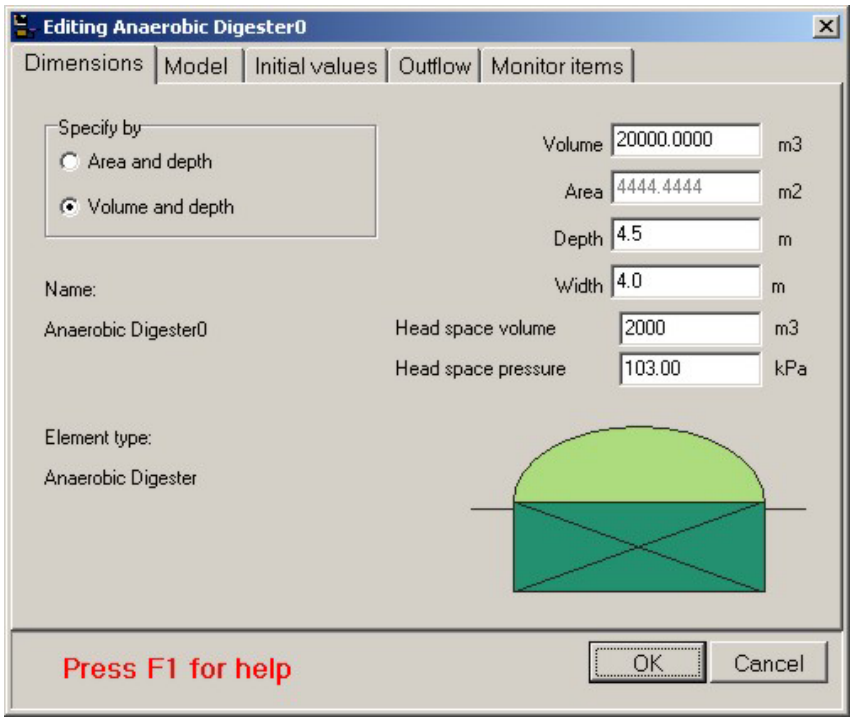
The anaerobic digester simulates the anaerobic digestion process in a CSTR. You may specify parameters related to the operation and control of the anaerobic digester element. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The anaerobic digester monitor items tab

Anaerobic Digester Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of an anaerobic digester element.



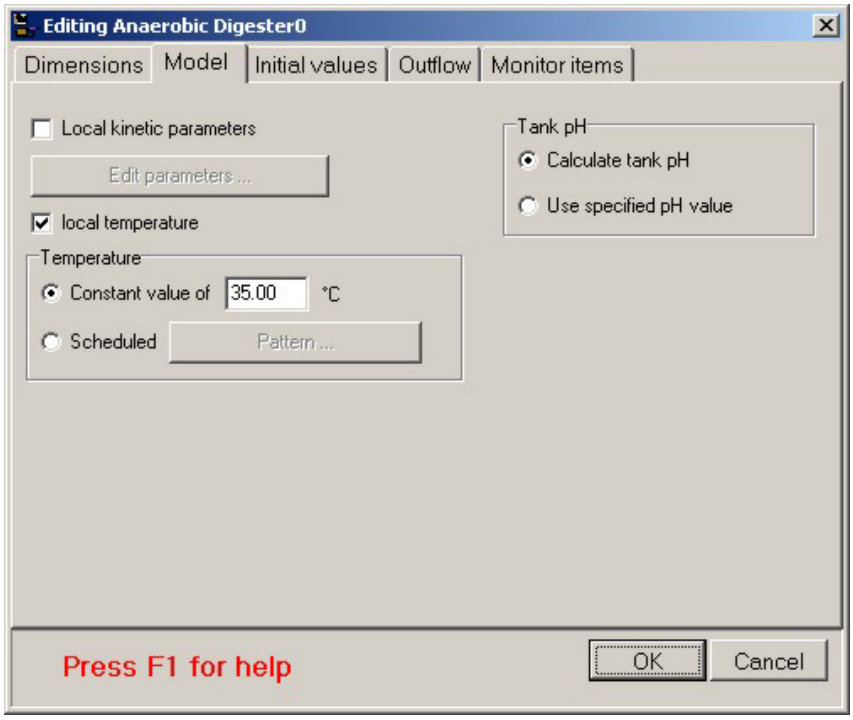
The anaerobic digester dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the anaerobic digester area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the anaerobic digester volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a **Width** for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

You must also enter the **Head space volume** and the **Head space pressure** for the anaerobic digester.

Anaerobic Digester Model

The **Model** tab, shown below, allows the user to specify parameters impacting the model used in the anaerobic digester element (such as local temperature) and also specify local model parameters.



The anaerobic digester model tab

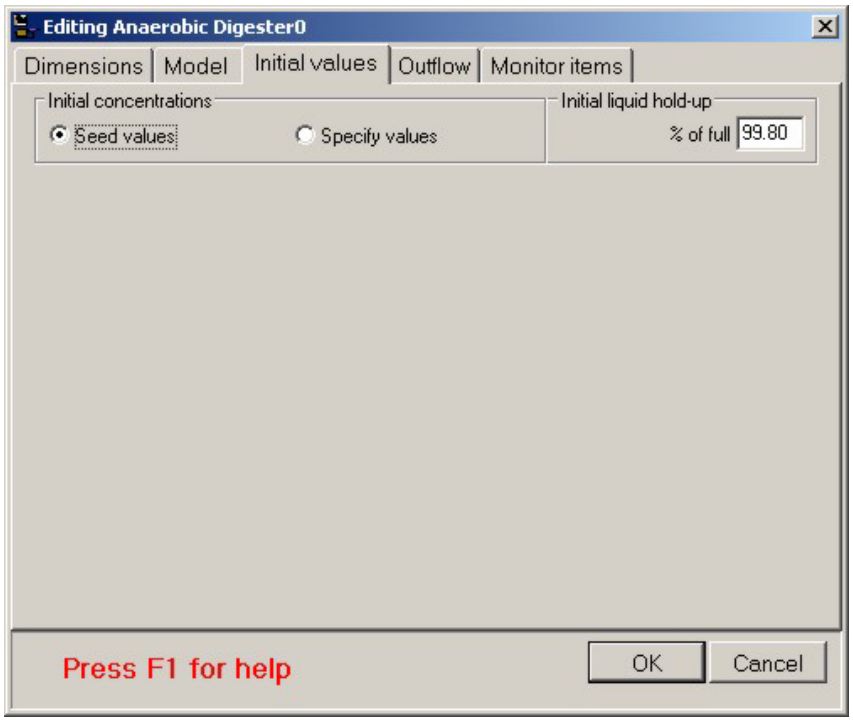
You can select **Local kinetic parameters** and click the **Edit parameters…** button in order to specify kinetic parameters for the anaerobic digester that are different from the configuration global parameters.

You can select whether you want BioWin to **Calculate tank pH** or you can **Use a specified value** if you wish.

A **Local temperature** may be specified for an anaerobic digester element. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

Anaerobic Digester Initial Values

The **Initial values** tab, shown below, is used for specifying the initial settings for anaerobic digester element concentrations and volume.

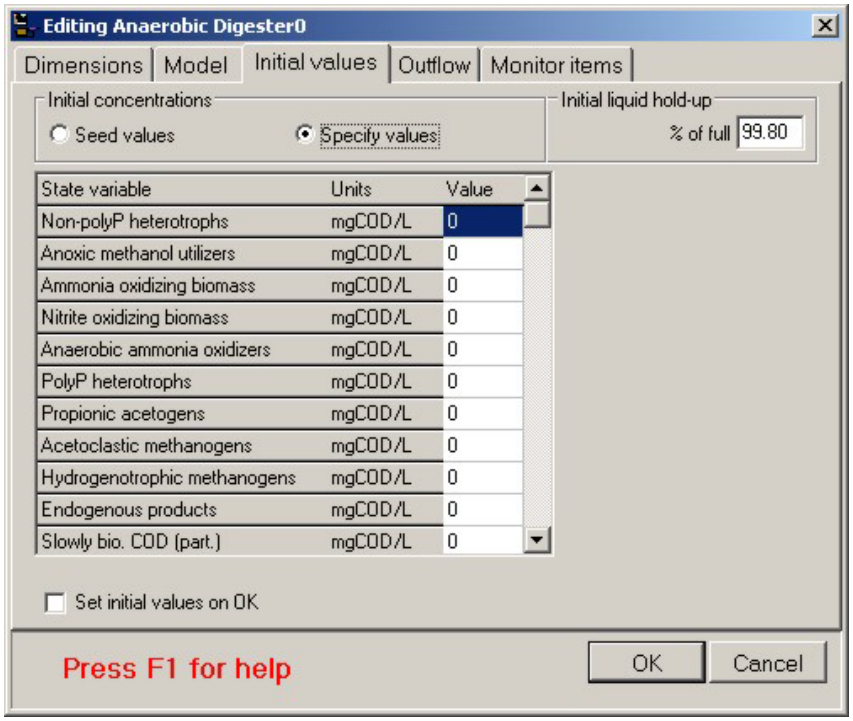


The anaerobic digester initial values tab with the Seed values *option active*

Use the **Initial liquid hold-up** field to enter a value to specify the **% of full** setting. This initial liquid hold-up volume will be used for steady state calculations. It should be noted here that the lower and upper limits on this value are 0.02 and 99.98%, respectively. This reflects the fact that this is the liquid volume, and allows for small differences between *liquid* volume and *reactor* volume for inlet and outlet pipes.

There are two methods for setting up **Initial concentrations** in the anaerobic digester element. The dialog box shown above illustrates the first case where the **Seed values** option is selected. When this option is selected, BioWin applies default seed values for all the state variables (except volume, which you specify) in the same manner it would for bioreactor elements.

The dialog box shown below illustrates the second case where the **Specify values** option is selected. In this case, an editable list of state variables is displayed in the dialog box. This allows you to enter your own seed values for state variables in the anaerobic digester element. These will be inserted in the anaerobic digester element when you begin a steady state simulation, or when you begin a dynamic simulation and choose to use seed values.



The anaerobic digester element initial values tab with the Specify values *option active*

When the specified seed values get inserted in the anaerobic digester element is determined by the **Set initial values on OK** option. If this option **is not** checked:

- The specified seed values will be placed in the anaerobic digester element at the beginning of a steady state simulation if you elect to start from seed conditions;
- The specified values will be placed in the anaerobic digester element at the beginning of a dynamic simulation **if** you elect to seed the simulation.

If this option **is** checked, the specified values are placed in the anaerobic digester element when you click **OK** to close the dialog box, **overriding any existing state variable values**. Therefore, the state variable values at the beginning of a dynamic simulation will be the same regardless of whether or not you choose to re-seed.

The use of this option is illustrated in the example at the end of the **Variable Volume/Batch Bioreactor Initial Values** section in this chapter.

Anaerobic Digester Outflow

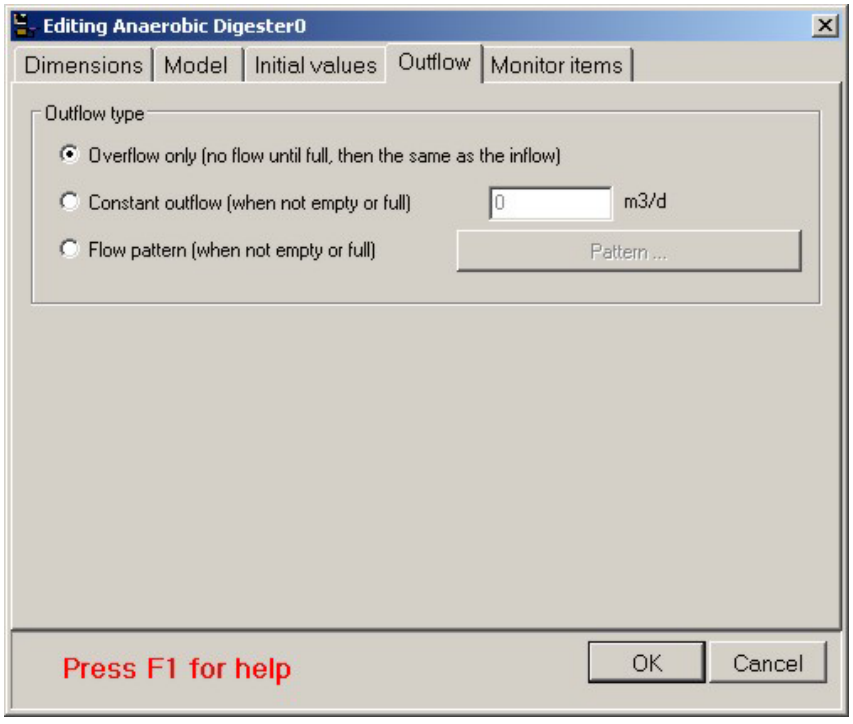
Remember for steady state simulations, the initial liquid hold-up volume will be used for calculations.

The anaerobic digester element **Outflow** tab, shown below, is used to specify the overflow behavior. The overflow behavior can be generalized as follows:

- Whenever the anaerobic digester element is full, it overflows at the influent rate, regardless of the overflow setting.
- Whenever the anaerobic digester element is empty and the outflow rate is set higher than the influent rate, the anaerobic digester element will only have an outflow equal to the influent flow, so as not to have negative volume. If the outflow rate is set lower than the influent rate, then the anaerobic digester element will begin to fill up.

The following is a description of the behavior of the three outflow settings:

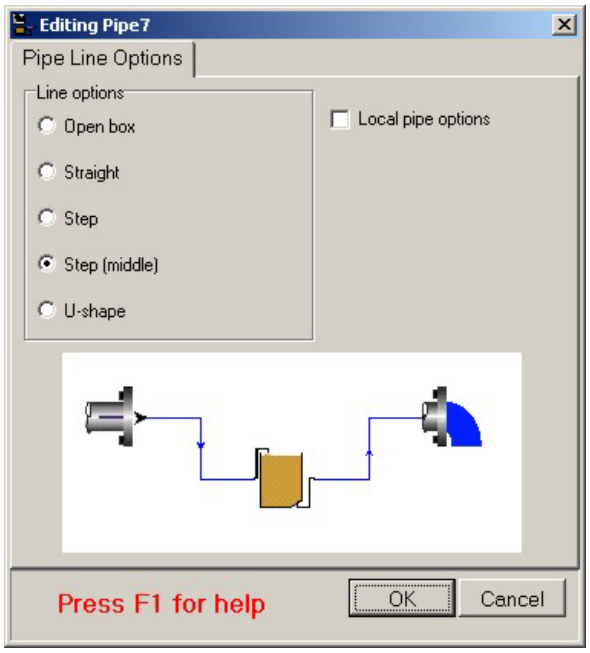
- **Overflow only** – the anaerobic digester element fills up, and then overflows at the influent flow rate.
- **Constant outflow** – the outflow always tries to attain the specified constant rate, except when physically constrained (i.e. when the anaerobic digester element is full or empty)
- **Flow pattern** – the outflow always tries to attain the current specified pattern rate, except when physically constrained (i.e. when the anaerobic digester element is full or empty). To specify a pattern, click the **Pattern…** button when it becomes active. For more information , see the **Liquid outflow itinerary** section.



The anaerobic digester outflow tab

Pipes

The **Pipe line options** tab, shown below, allows the user to specify pipe display options. You can access this dialog box by double-clicking a pipe on the drawing board or right-clicking a pipe and choosing **Properties...** from the resulting popup menu.



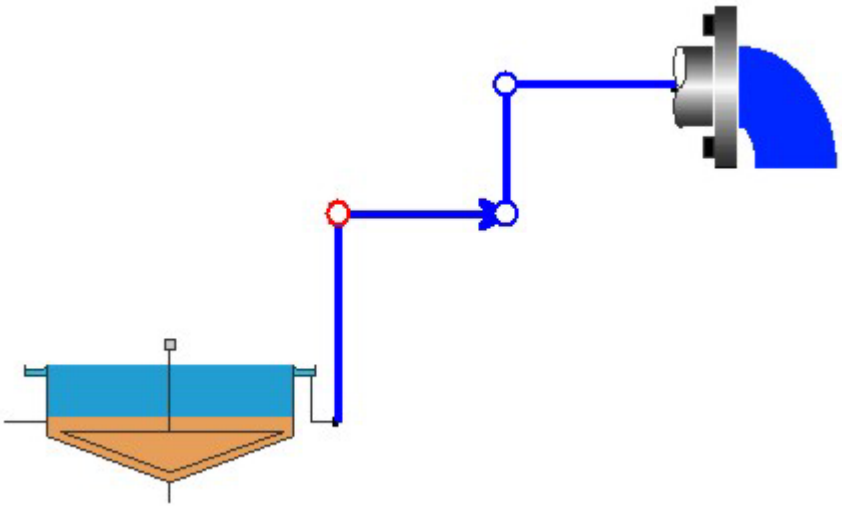
The pipe line options tab

There are five pipe styles from which you can select by clicking the appropriate radio button:

1. Open box;
2. Straight;
3. Step;
4. Step (middle) – this is the default pipe style;
5. U-shape

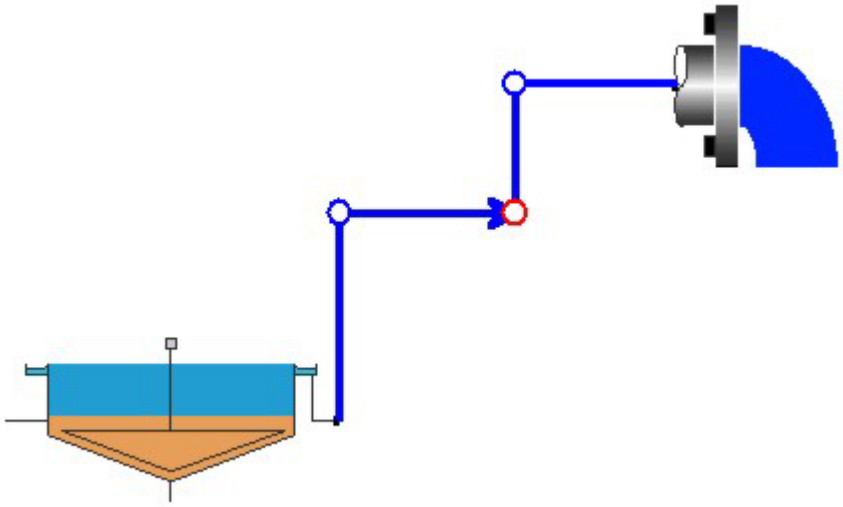
An example of the selected option is shown in the picture on the lower part of the tab. Pipes must start from and end at an element. Clicking once on a pipe will reveal that it has "tag" points (denoted by small circles on the pipe). If a circle is red, then it may be moved. The various pipe styles differ in how these points may be moved by dragging them around the drawing board.

The **Open Box** style has two points which may be moved. This makes the **Open Box** style the most flexible of the available styles. This flexibility is useful in avoiding pipes crossing over one another and going through elements in complex configurations with internal recycles. The first point (i.e. the point closest to the pipe's beginning) is free to move up and down in the vertical direction, as shown in the picture below.



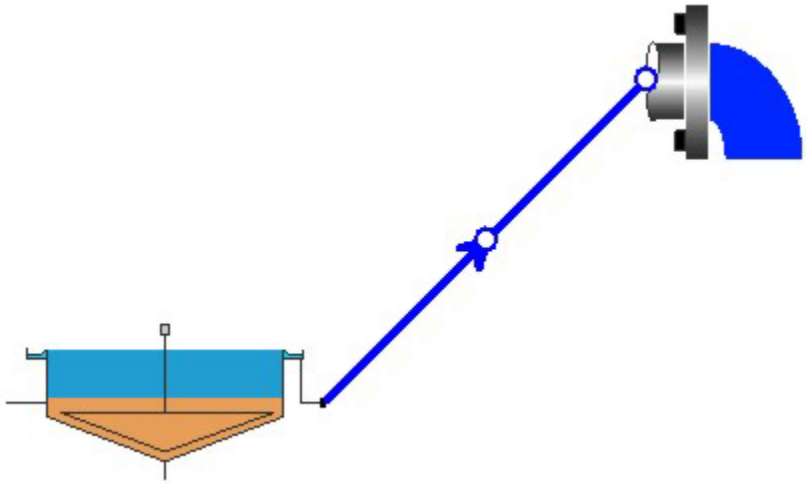
The first point for an open box pipe can move in the vertical direction

The second point (i.e. the middle point of the three) is free to move about in the horizontal direction, as shown in the picture below.



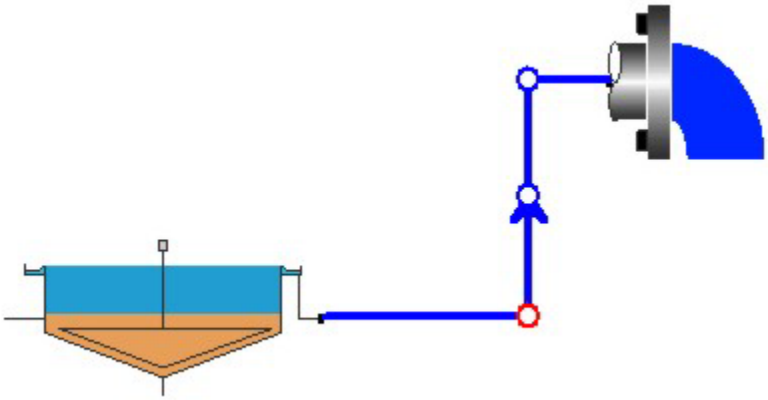
The second point for an open box can move in the horizontal direction

The **Straight** style has two points as shown in the picture below. Neither of these points may be moved – the pipe simply is a straight line drawn between the two elements.



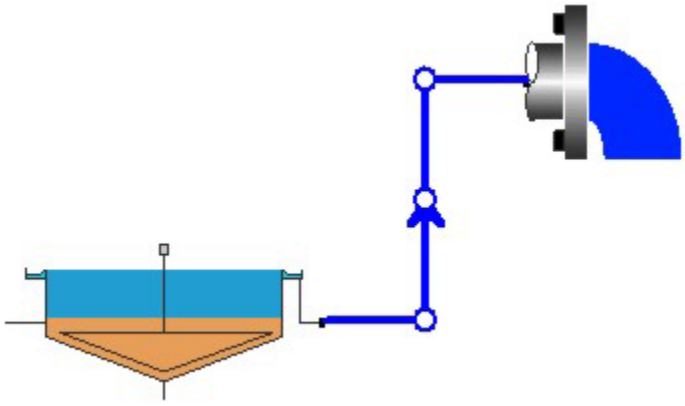
A straight pipe's points may not be moved about the drawing board

The **Step** style has one point which may be moved. The first point (i.e. the point closest to the pipe's beginning) may be moved about in the horizontal direction, as shown in the picture below.



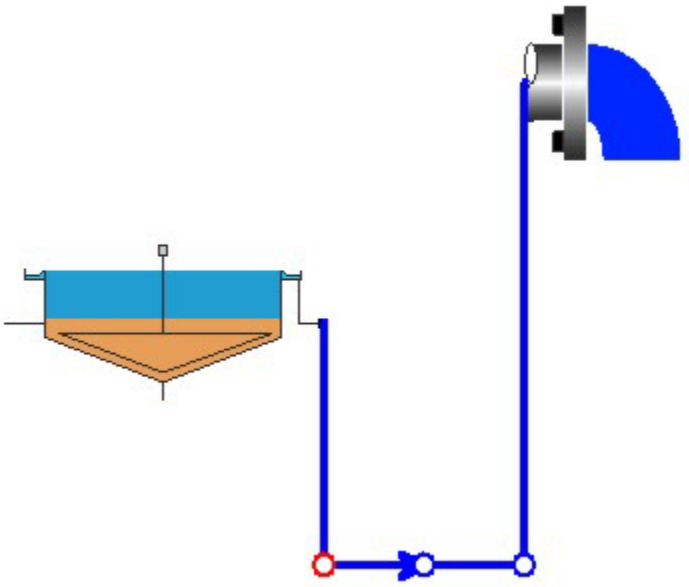
A step pipe has one point which may be moved horizontally

The **Step (middle)** style has three points as shown in the picture below. None of these points may be moved – the pipe automatically forms a step at the point halfway between the elements.



A step (middle) pipe has no moveable points

The **U-shape** style has one point which may be moved. The first point (i.e. the point closest to the pipe's beginning) may be moved about in the vertical direction, as shown in the picture below.



A U-shape has one point which may be moved vertically

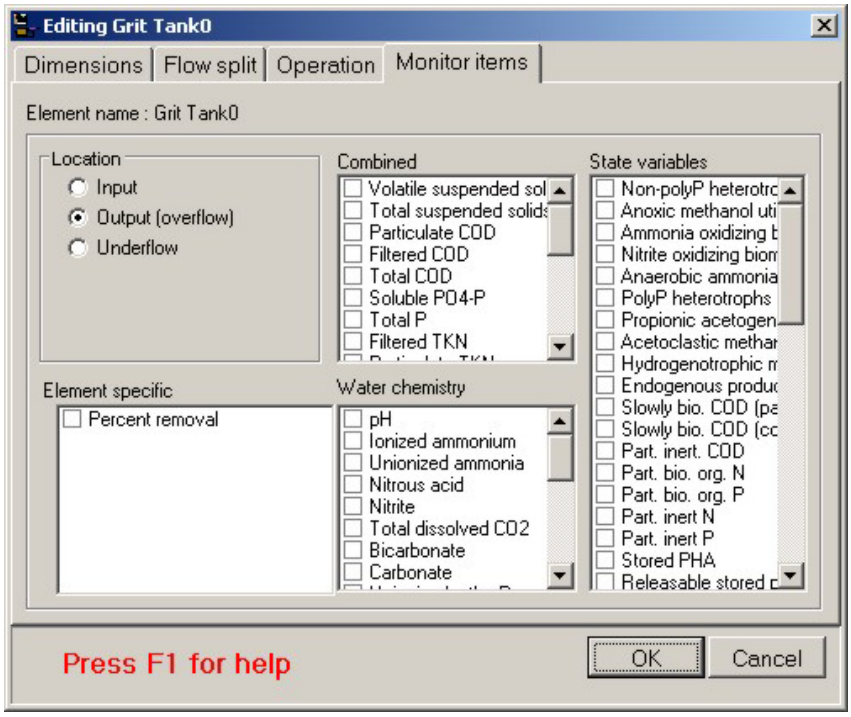
You can change line or arrow properties for individual pipes so that they are different from the settings you have chosen in **Project|Current Project Options…** by clicking on the check box labeled **Local pipe options**; a **Line options…** button and **Arrow size** and **Arrow angle** spin edit boxes will appear.

The "arrow angle" refers to the acute angle between the arrow side and the pipe line. For example, if you want arrows with "flat" bases, set the arrow side angle to the maximum of 60 degrees.

To change the **Color**, **Width**, and **Style** of the lines used to represent pipes on the drawing board, click on the **Line options…** button. You may increase the **arrow size** on the lines used to represent pipes. The **arrow angle** also can be changed.

Grit Removal Tank

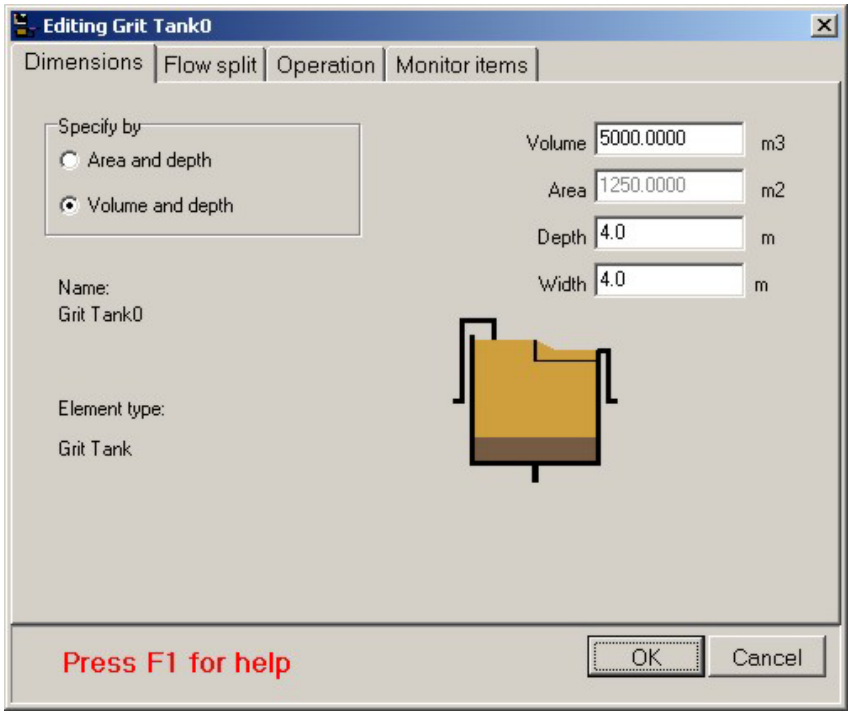
The grit removal tank element is used to simulate the removal of sand and other inerts in a system. You may specify details regarding the physical specifications, flow split method, and solids separation operation of the grit tank. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The grit removal tank monitor items tab

Grit Removal Tank Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of a grit removal tank element.

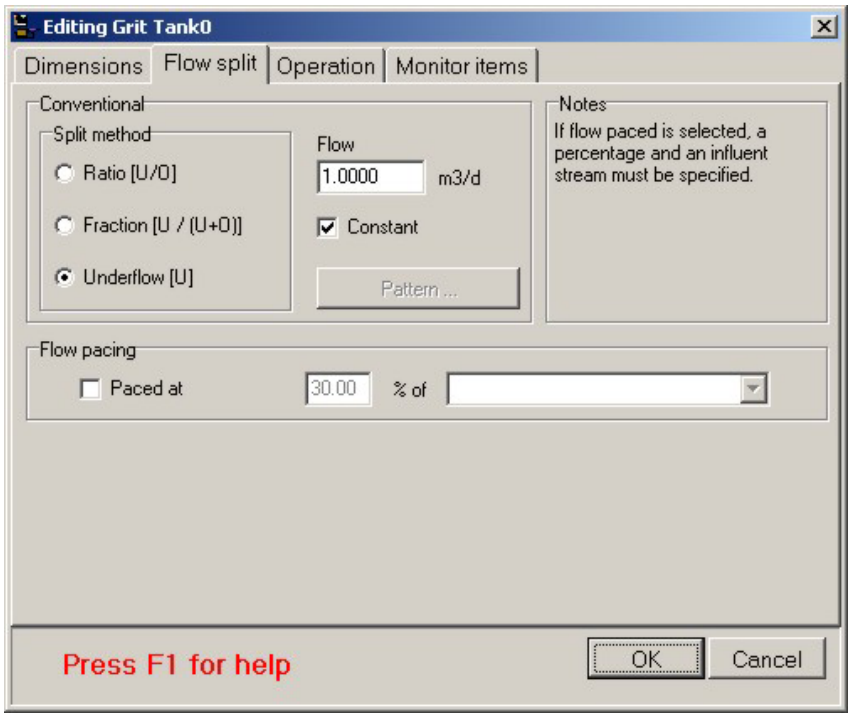


The grit removal tank dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the grit removal tank area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the grit removal tank volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a **Width** for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

Grit Removal Tank Split Method

The **Flow split** tab, shown below, allows the user to specify the flow split method for a grit removal tank.



The grit removal tank split method tab

The method of specifying the flow split for a grit removal tank may be selected from a number of options. You can specify the flow using a **Ratio**, **Fraction**, or **Underflow rate** by clicking on the corresponding radio button. If you specify an underflow rate (denoted by the symbol **U**), it will result in a constant underflow out the bottom of the grit removal tank. Note that when the grit removal tank is operating in this mode, if the influent flow is less than the set **Underflow rate**, then all of the influent flow will be sent to the underflow. If you specify either the **Ratio** or **Fraction** split method, then the underflow will be calculated using the corresponding formula based on the overflow rate (denoted by the symbol **O**) and the underflow rate **U**.

Next to the split method radio group is an edit box; if the split method is selected from the radio group, the value may be entered in the edit box. Units are shown adjacent to the edit box (where applicable) and the label of the edit box changes to indicate the chosen split method.

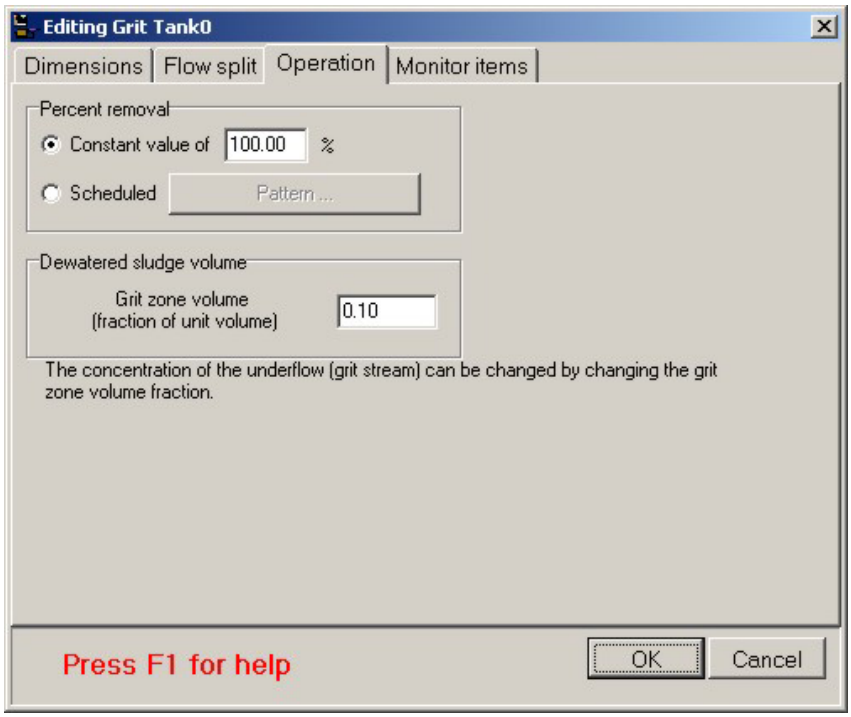
The split method group also contains a **Constant** check box; if the flow split does not vary with time this box should be checked. It should be noted that this check box

applies only to the split methods listed in the radio group and is checked by default. If the **Constant** check box is unchecked, the **Pattern…** button becomes active so you can enter a time-varying pattern for the flow split. Clicking this button presents you with the **Edit split itinerary** dialog box.

The underflow rate also can be **Paced** with an influent stream. To select the flow paced option, click on the **Paced at** check box. The percentage of the influent flow rate may then be specified, and the influent stream for flow pacing may be selected from the drop list box which shows all influent streams available for your system.

Grit Removal Tank Operation

The **Operation** tab, shown below, allows you to specify the amount of grit removed by the grit removal tank element.



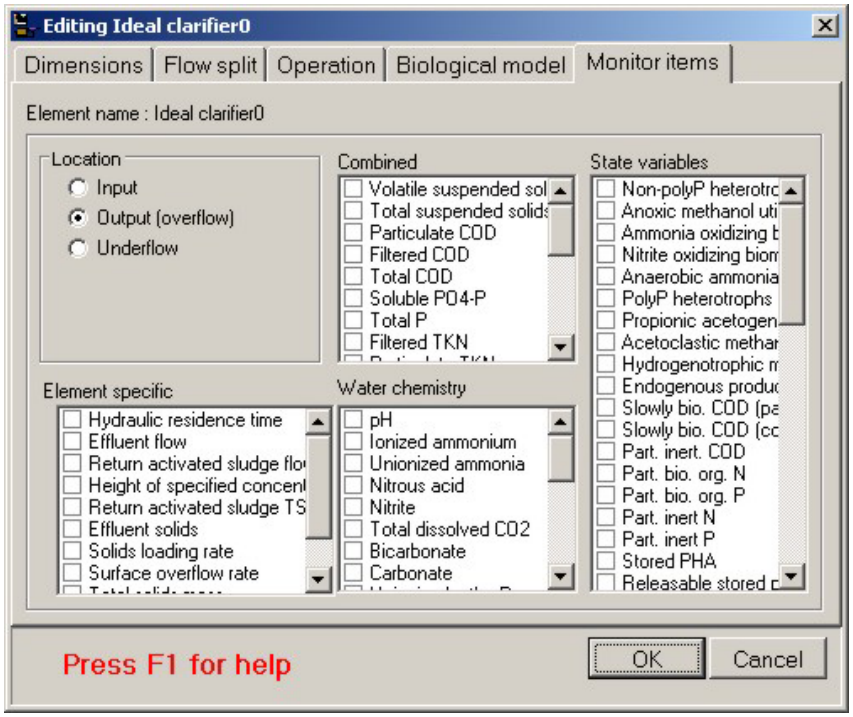
The grit removal tank operation tab

There are two edit boxes on this tab. The **Percentage grit removal** may be entered in the first edit box. This is the average percentage grit removal, and may differ from the instantaneous percentage removal observed in a dynamic simulation.

The second edit box is provided for specifying the **Grit zone volume fraction**; this is the grit zone fraction of a unit volume. The concentration of the underflow (grit stream) can be modified by adjusting this fraction.

Ideal Primary Settler

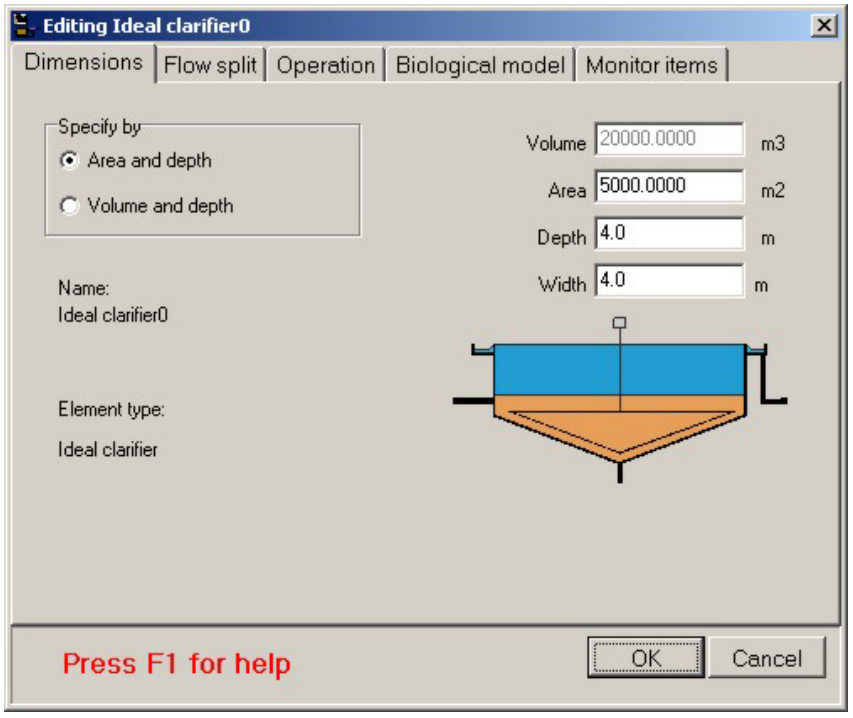
The ideal primary settler (clarifier) element is used to model settlement of particulate material in a wastewater stream which does not contain activated sludge mixed liquor (e.g. a raw influent stream). You may specify the physical characteristics, the flow split method, and the solids separation operation of the primary settler. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The ideal primary settling tank monitor items tab

Ideal Primary Settler Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of an ideal primary settler element.

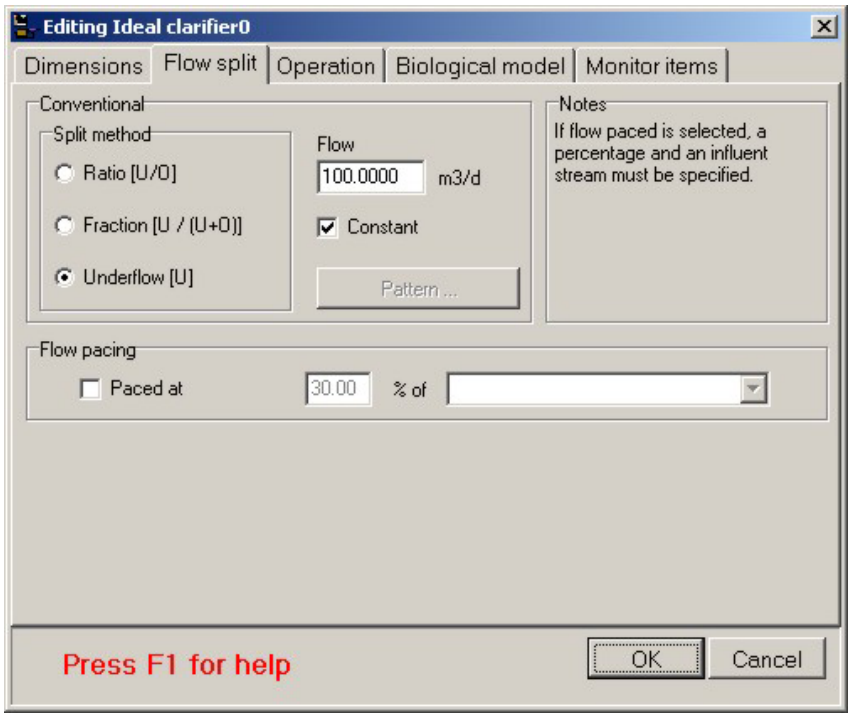


The ideal primary settler dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the ideal primary settler area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the ideal primary settler volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a **Width** for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

Ideal Primary Settler Split Method

The **Flow split** tab, shown below, allows the user to specify the flow split method for an ideal primary settler.



The ideal primary settler flow split method tab

The method of specifying the flow split for an ideal primary settler may be selected from a number of options. You can specify the flow using a **Ratio**, **Fraction**, or **Underflow rate** by clicking on the corresponding radio button. If you specify an underflow rate (denoted by the symbol **U**), it will result in a constant underflow out the bottom of the ideal primary settler. Note that when the ideal primary settler is operating in this mode, if the influent flow is less than the set **Underflow rate**, then all of the influent flow will be sent to the underflow. If you specify either the **Ratio** or **Fraction** split method, then the underflow will be calculated using the corresponding formula based on the overflow rate (denoted by the symbol **O**) and the underflow rate **U**.

Next to the split method radio group is an edit box; if the split method is selected from the radio group, the value may be entered in the edit box. Units are shown adjacent to the edit box (where applicable) and the label of the edit box changes to indicate the chosen split method.

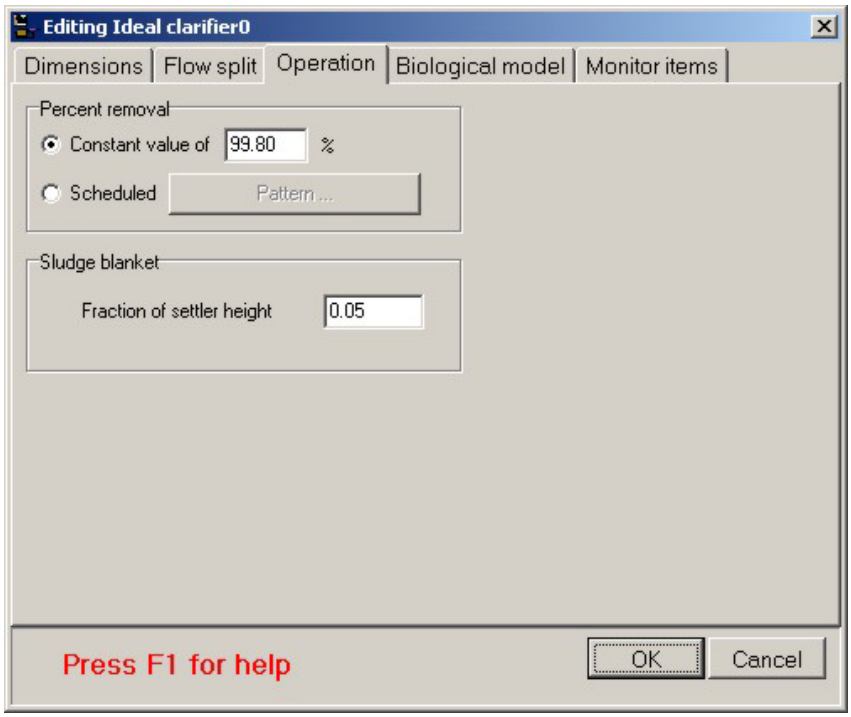
The split method group also contains a **Constant** check box; if the flow split does not vary with time this box should be checked. It should be noted that this check box

applies only to the split methods listed in the radio group and is checked by default. If the **Constant** check box is unchecked, the **Pattern…** button becomes active so you can enter a time-varying pattern for the flow split. Clicking this button presents you with the **Edit split itinerary** dialog box.

The underflow rate also can be **Paced** with an influent stream. To select the flow paced option, click on the **Paced at** check box. The percentage of the influent flow rate may then be specified, and the influent stream for flow pacing may be selected from the drop list box which shows all influent streams available for your system.

Ideal Primary Settler Operation

The **Operation** tab, shown below, allows the user to enter configuration specifications for an ideal primary settling tank.



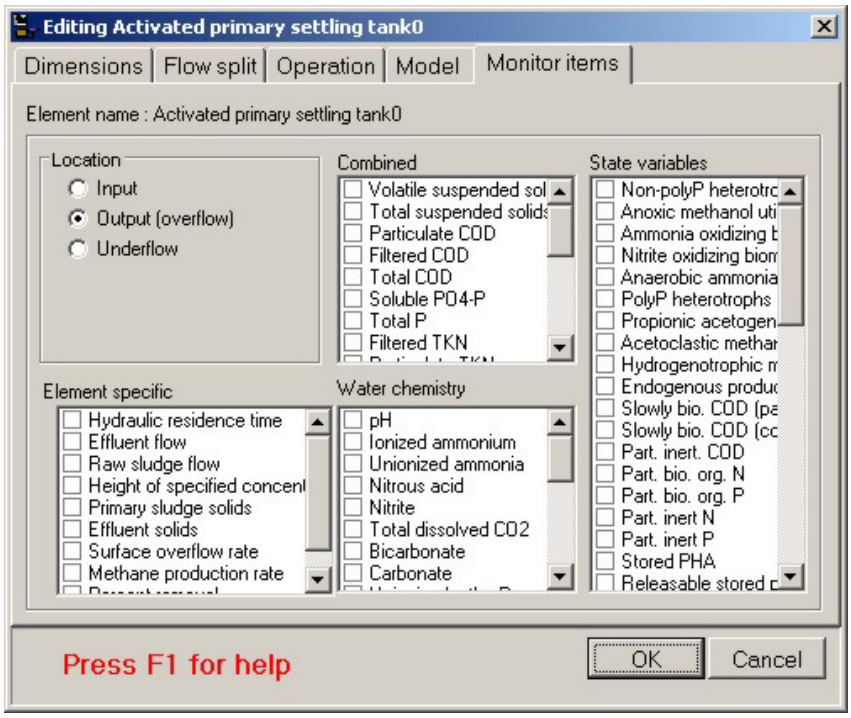
The ideal primary settler operation tab

There are two edit boxes on this tab. The first edit box is used to enter the **Percentage solids removal** for the ideal primary settler; the **Sludge blanket height** may be specified using the second box. Sludge blanket height is expressed as a fraction of the total settler height

Activated Primary Settler

The activated primary settler (clarifier) element is used to model settlement of particulate material in a wastewater stream which does not contain activated sludge mixed liquor (e.g. a raw influent stream). The aspect which differentiates the activated primary settler element from the ideal primary settler element is the former is capable of modeling biological activity which may take place in a primary settler. You may specify the physical characteristics, the flow split method, the solids separation operation, and the model details of the activated primary settler. For

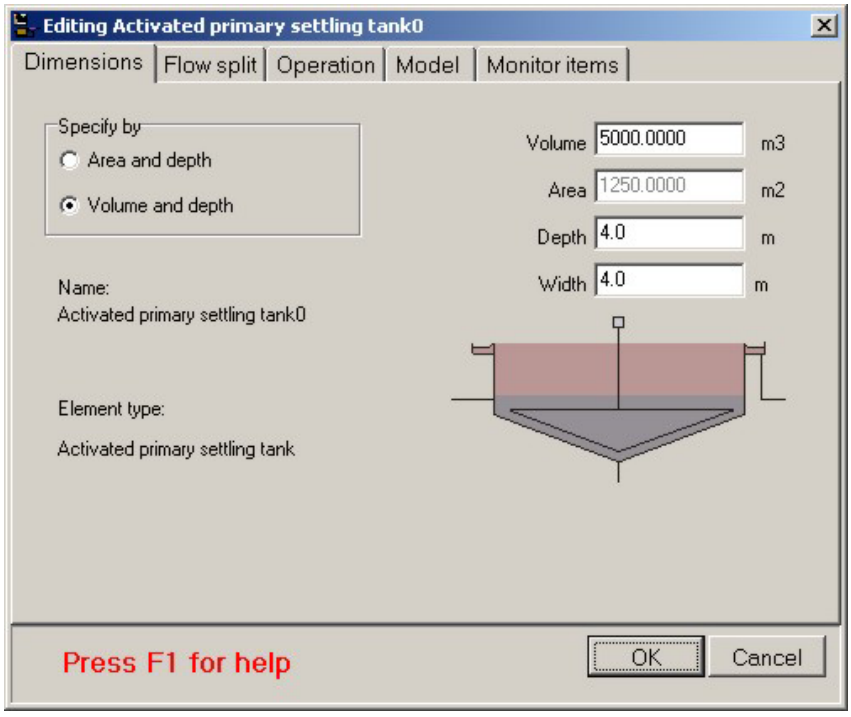
information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The activated primary settling tank monitor items tab

Activated Primary Settler Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of an activated primary settler element.

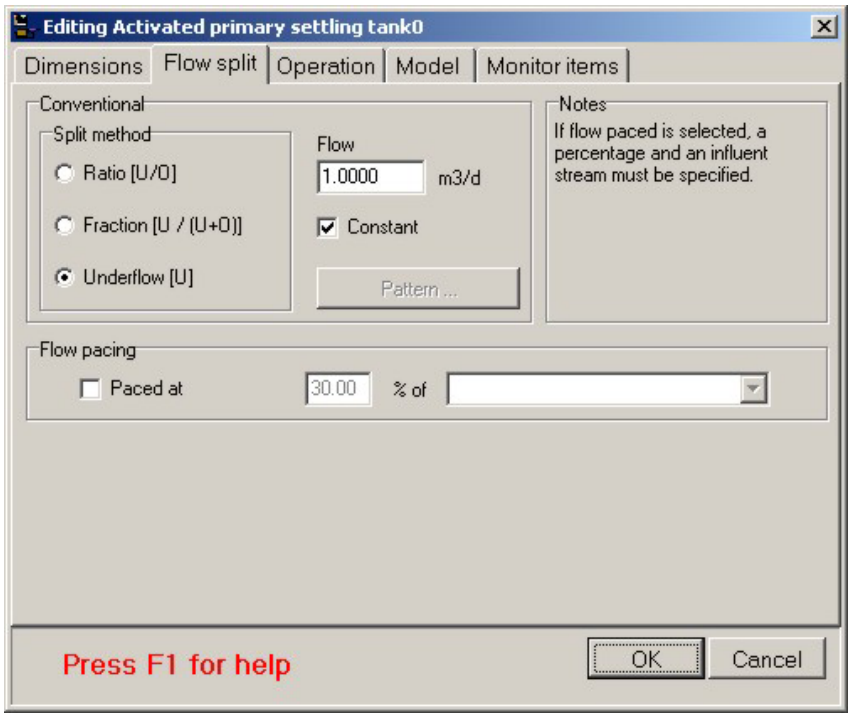


The activated primary settler dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the activated primary settler area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the activated primary settler volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a **Width** for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

Activated Primary Settler Split Method

The **Flow split** tab, shown below, allows the user to specify the flow split method for an activated primary settler.



The activated primary settler flow split method tab

The method of specifying the flow split for an activated primary settler may be selected from a number of options. You can specify the flow using a **Ratio**, **Fraction**, or **Underflow rate** by clicking on the corresponding radio button. If you specify an underflow rate (denoted by the symbol **U**), it will result in a constant underflow out the bottom of the activated primary settler. Note that when the activated primary settler is operating in this mode, if the influent flow is less than the set **Underflow rate**, then all of the influent flow will be sent to the underflow. If you specify either the **Ratio** or **Fraction** split method, then the underflow will be calculated using the corresponding formula based on the overflow rate (denoted by the symbol **O**) and the underflow rate **U**.

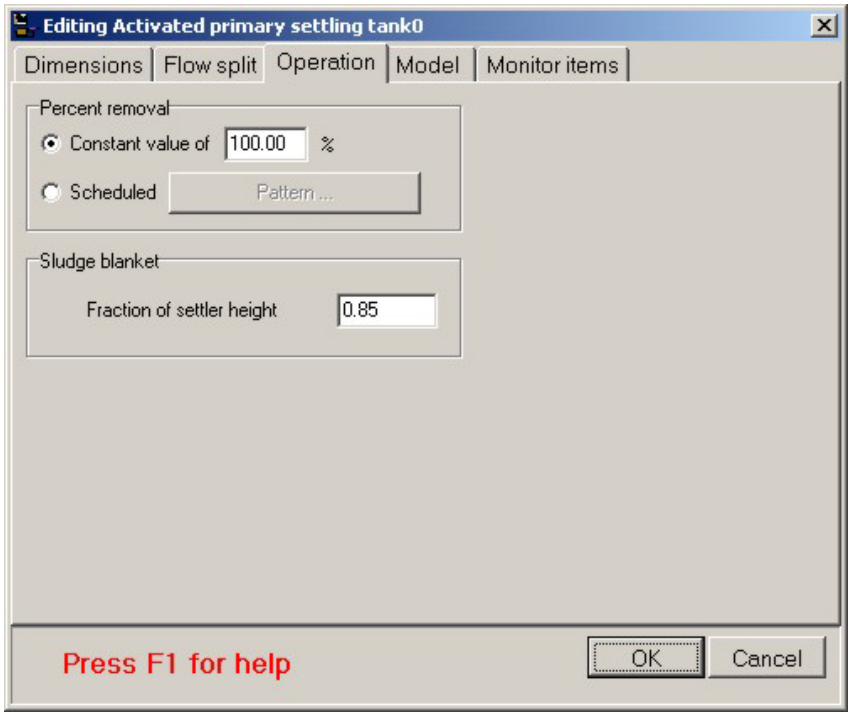
Next to the split method radio group is an edit box; if the split method is selected from the radio group, the value may be entered in the edit box. Units are shown adjacent to the edit box (where applicable) and the label of the edit box changes to indicate the chosen split method.

The split method group also contains a **Constant** check box; if the flow split does not vary with time this box should be checked. It should be noted that this check box applies only to the split methods listed in the radio group and is checked by default. If the **Constant** check box is unchecked, the **Pattern…** button becomes active so you can enter a time-varying pattern for the flow split. Clicking this button presents you with the **Edit split itinerary** dialog box..

The underflow rate also can be **Paced** with an influent stream. To select the flow paced option, click on the **Paced at** check box. The percentage of the influent flow rate may then be specified, and the influent stream for flow pacing may be selected from the drop list box which shows all influent streams available for your system.

Activated Primary Settler Operation

The **Operation** tab, shown below, allows the user to enter configuration specifications for an activated primary settling tank.

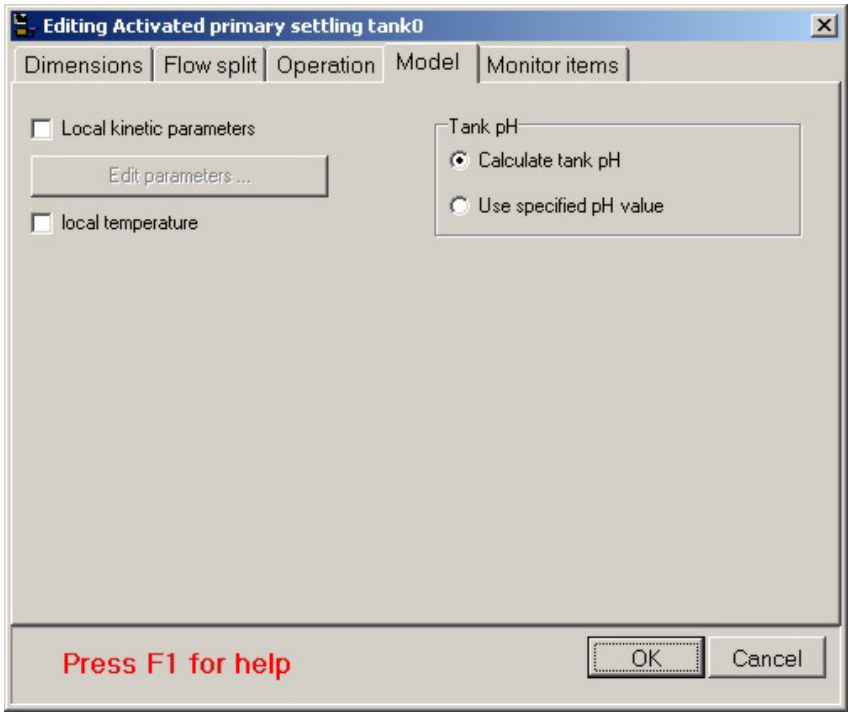


The activated primary settler operation tab

There are two edit boxes on this tab. The first edit box is used to enter the **Percentage solids removal** for the activated primary settler; the **Sludge blanket height** may be specified using the second box. Sludge blanket height is expressed as a fraction of the total settler height.

Activated Primary Settler Model

The **Model** tab, shown below, allows the user to specify parameters impacting the model used in the activated primary settler element (such as local temperature) and also specify local model parameters.



The activated primary model tab

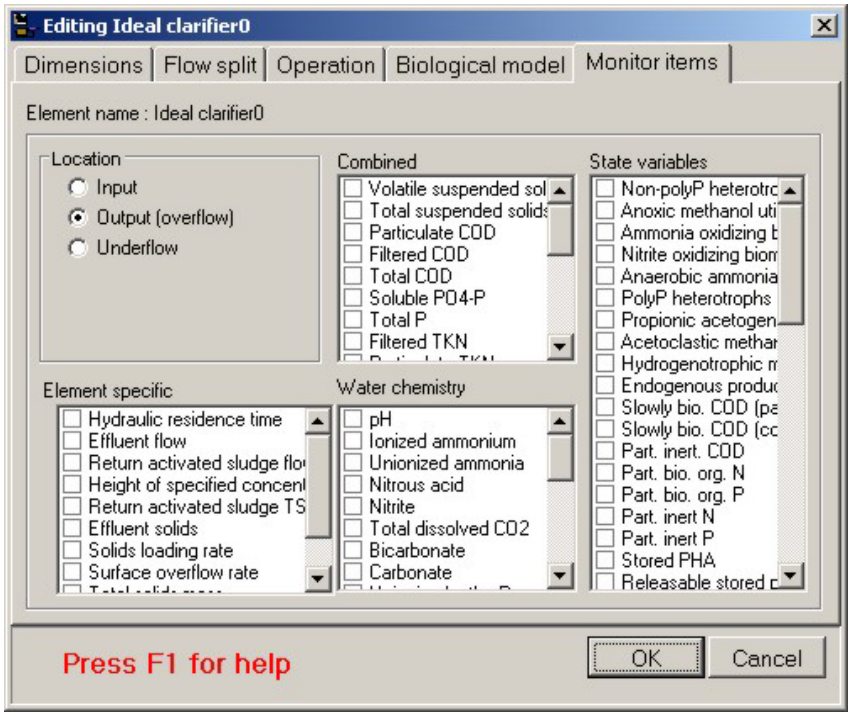
You can select **Local kinetic parameters** and click the **Edit parameters…** button in order to specify kinetic parameters for the activated primary that are different from the configuration global parameters.

You can select whether you want BioWin to **Calculate tank pH** or you can **Use a specified value** if you wish.

A **Local temperature** may be specified for an activated primary settler element. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

Ideal Secondary Settler

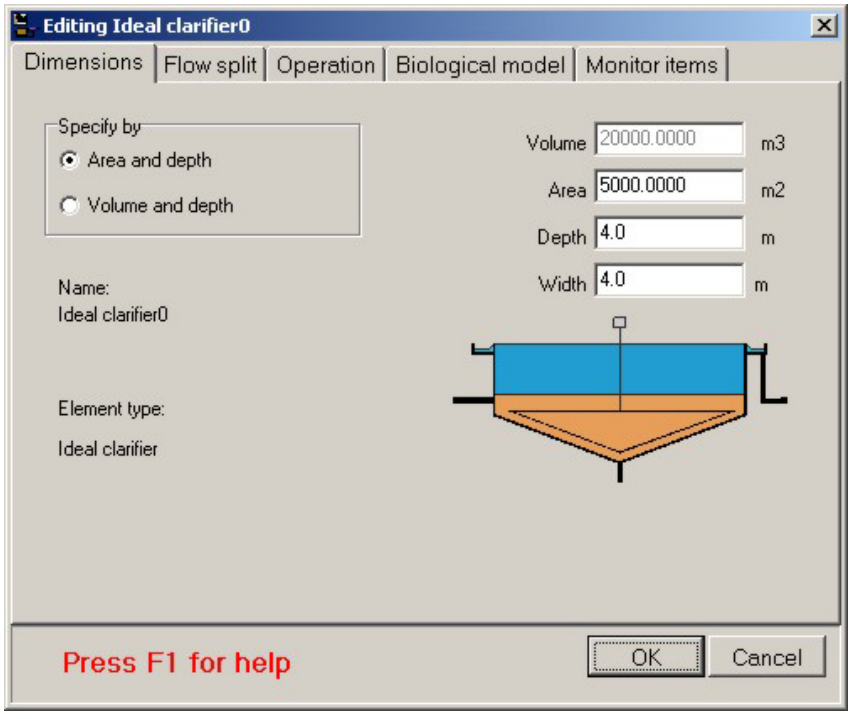
The ideal secondary settler (clarifier) element is used to model settlement of particulate material in a wastewater stream containing activated sludge mixed liquor based on an idealized solids separation model. You may specify the physical characteristics, the flow split method, the solids separation operation, and biological reaction of the ideal secondary settler. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The ideal secondary settler monitor items tab

Ideal Secondary Settler Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of an ideal secondary settler element.

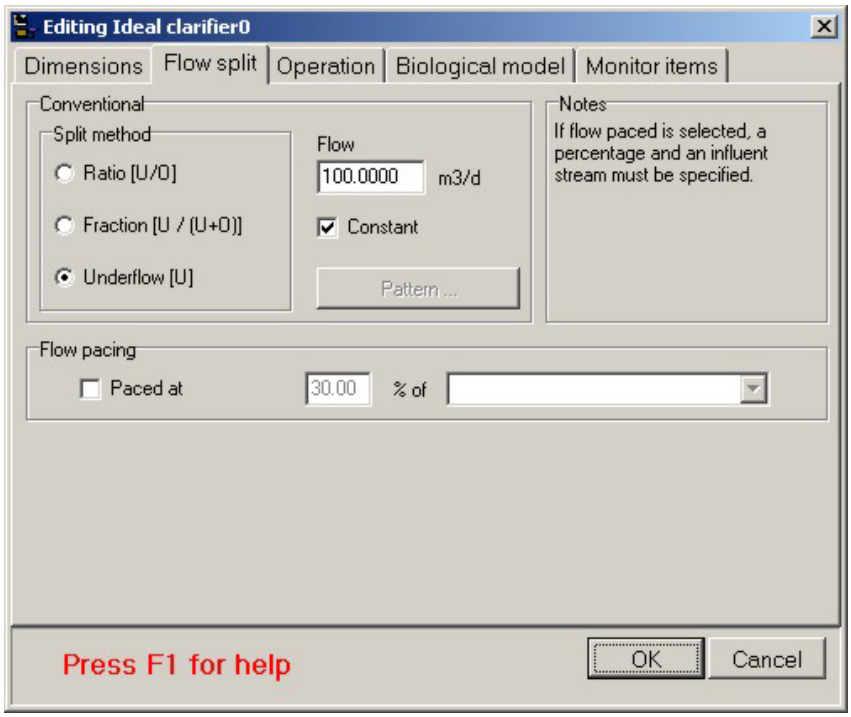


The ideal secondary settler dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the ideal secondary settler area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the ideal secondary settler volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a **Width** for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

Ideal Secondary Settler Split Method

The **Flow split** tab, shown below, allows the user to specify the flow split method for an ideal secondary settler.



The ideal secondary settler flow split method tab

The method of specifying the flow split for an ideal secondary settler may be selected from a number of options. You can specify the flow using a **Ratio**, **Fraction**, or **Underflow rate** by clicking on the corresponding radio button. If you specify an underflow rate (denoted by the symbol **U**), it will result in a constant underflow out the bottom of the ideal secondary settler. Note that when the ideal secondary settler is operating in this mode, if the influent flow is less than the set **Underflow rate**, then all of the influent flow will be sent to the underflow. If you specify either the **Ratio** or **Fraction** split method, then the underflow will be calculated using the corresponding formula based on the overflow rate (denoted by the symbol **O**) and the underflow rate **U**.

Next to the split method radio group is an edit box; if the split method is selected from the radio group, the value may be entered in the edit box. Units are shown adjacent to the edit box (where applicable) and the label of the edit box changes to indicate the chosen split method.

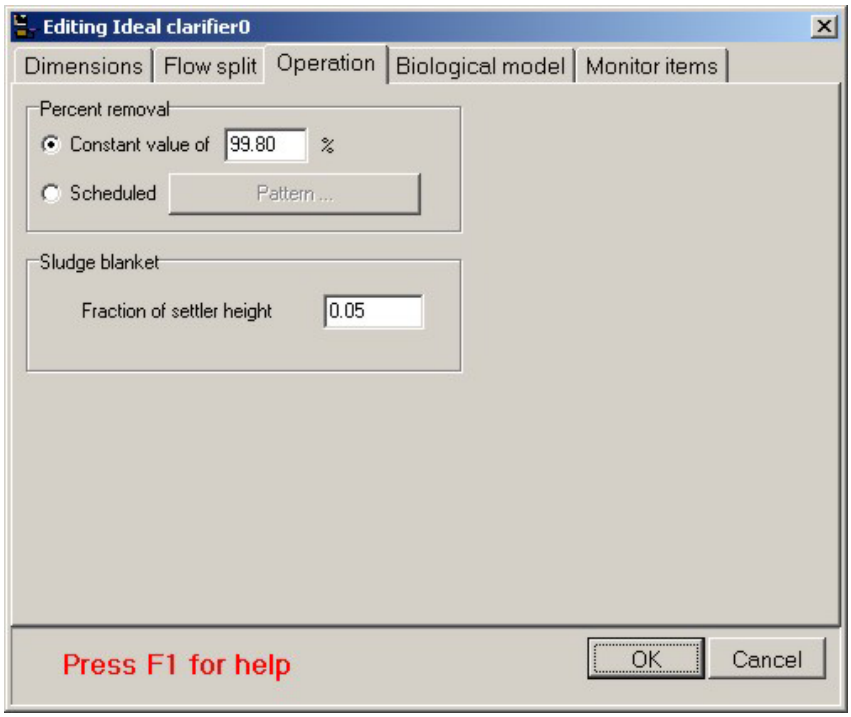
The split method group also contains a **Constant** check box; if the flow split does not vary with time this box should be checked. It should be noted that this check box

applies only to the split methods listed in the radio group and is checked by default. If the **Constant** check box is unchecked, the **Pattern…** button becomes active so you can enter a time-varying pattern for the flow split. Clicking this button presents you with the **Edit split itinerary** dialog box.

The underflow rate also can be **Paced** with an influent stream. To select the flow paced option, click on the **Paced at** check box. The percentage of the influent flow rate may then be specified, and the influent stream for flow pacing may be selected from the drop list box which shows all influent streams available for your system.

Ideal Secondary Settler Operation

The **Operation** tab, shown below, allows the user to enter configuration specifications for an ideal secondary settling tank.

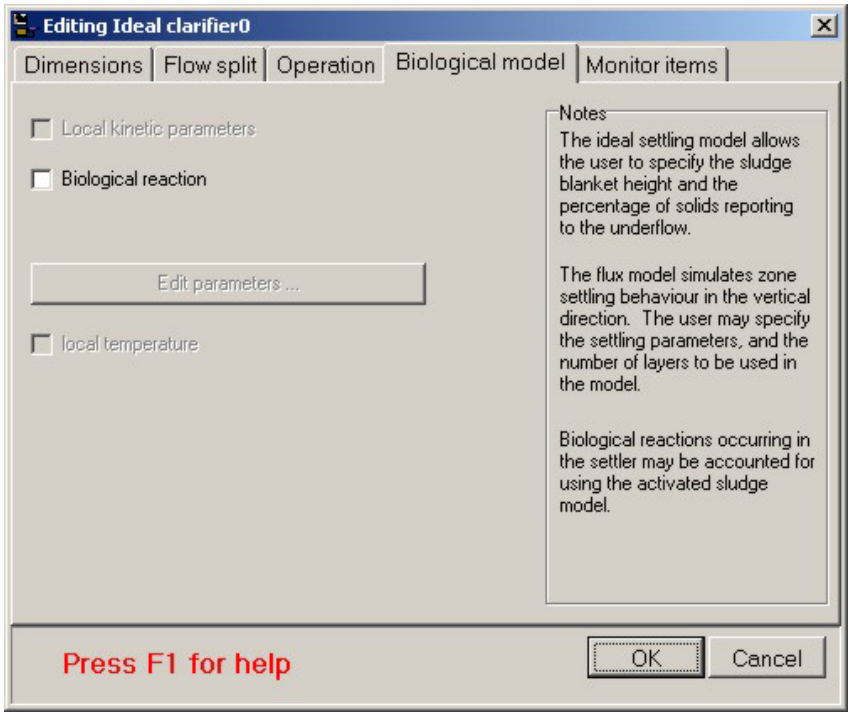


The ideal secondary settler operation tab

There are two edit boxes on this tab. The first edit box is used to enter the **Percentage solids removal** for the ideal secondary settler; the **Sludge blanket height** may be specified using the second box. Sludge blanket height is expressed as a fraction of the total settler height

Ideal Secondary Settler Biological Modeling

The ideal secondary settler **Biological model** tab, shown below, allows the user to specify whether biological reactions in the settler are modeled, and if so, model options such as local kinetic parameters and local temperature.



The ideal secondary settler biological modeling tab

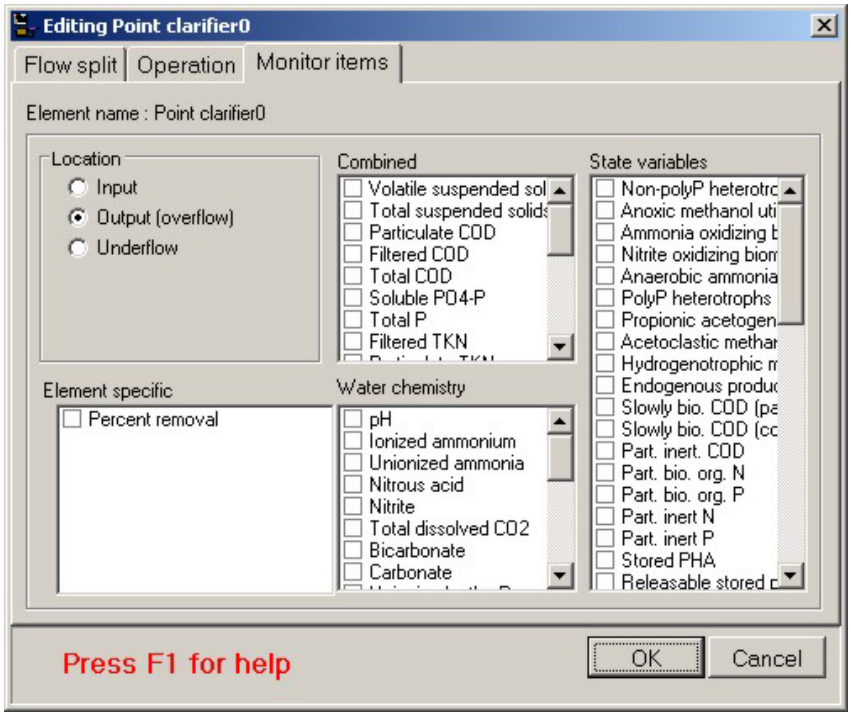
If you wish to model biological reaction in the ideal secondary settler, click on the check box for **Biological reaction**; this switches on the activated sludge processes and enables the **Local kinetic parameters** and **Local temperature** check boxes.

If you wish to specify **Local kinetic parameters** for the ideal secondary settler (i.e. kinetic parameters that are different from those specified under the **Project|Model parameters…** menu), click on the check box for local kinetic parameters, then click the **Model parameters…** button to open the **Model parameter editor** dialog box which allows you to access the various activated sludge model kinetic parameters.

A **Local temperature** also may be specified for an ideal secondary settler. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

Point Secondary Settler

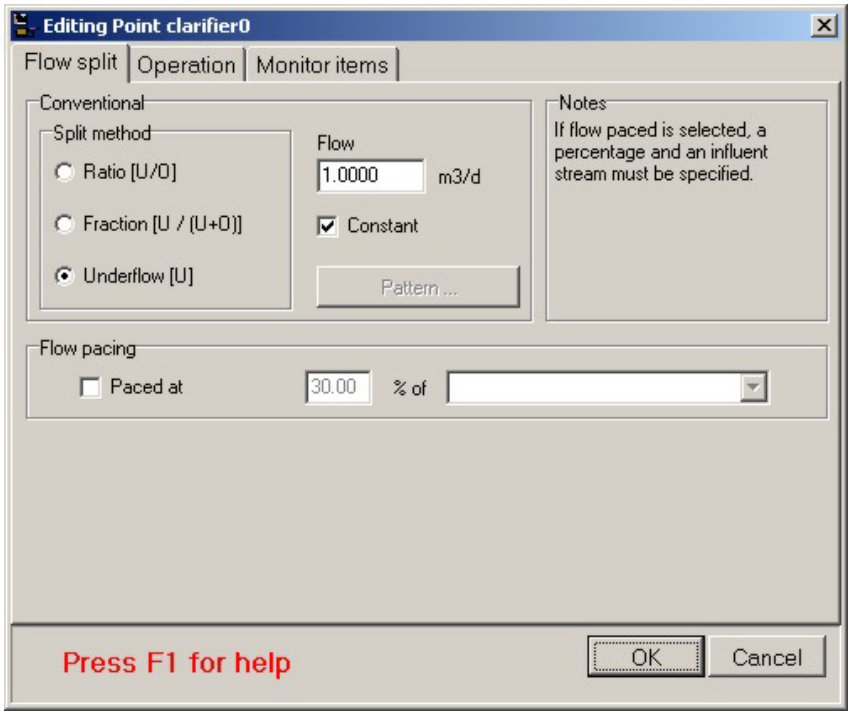
The point secondary settler (clarifier) element is similar to the ideal secondary settler element and is used to model settlement of particulate material in a wastewater stream containing activated sludge mixed liquor based on an idealized solids separation model. The distinctive feature of this element is that it has no volume. You may specify the flow split method and the solids removal percentage - because the element has no volume, biological reactions may not be modeled. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The point secondary settler monitor items tab

Point Secondary Settler Split Method

The **Flow split** tab, shown below, allows the user to specify the flow split method for a point secondary settler.



The point secondary settler flow split method tab

The method of specifying the flow split for a point secondary settler may be selected from a number of options. You can specify the flow using a **Ratio**, **Fraction**, or **Underflow rate** by clicking on the corresponding radio button. If you specify an underflow rate (denoted by the symbol **U**), it will result in a constant underflow out the bottom of the point secondary settler. Note that when the point secondary settler is operating in this mode, if the influent flow is less than the set **Underflow rate**, then all of the influent flow will be sent to the underflow. If you specify either the **Ratio** or **Fraction** split method, then the underflow will be calculated using the corresponding formula based on the overflow rate (denoted by the symbol **O**) and the underflow rate **U**.

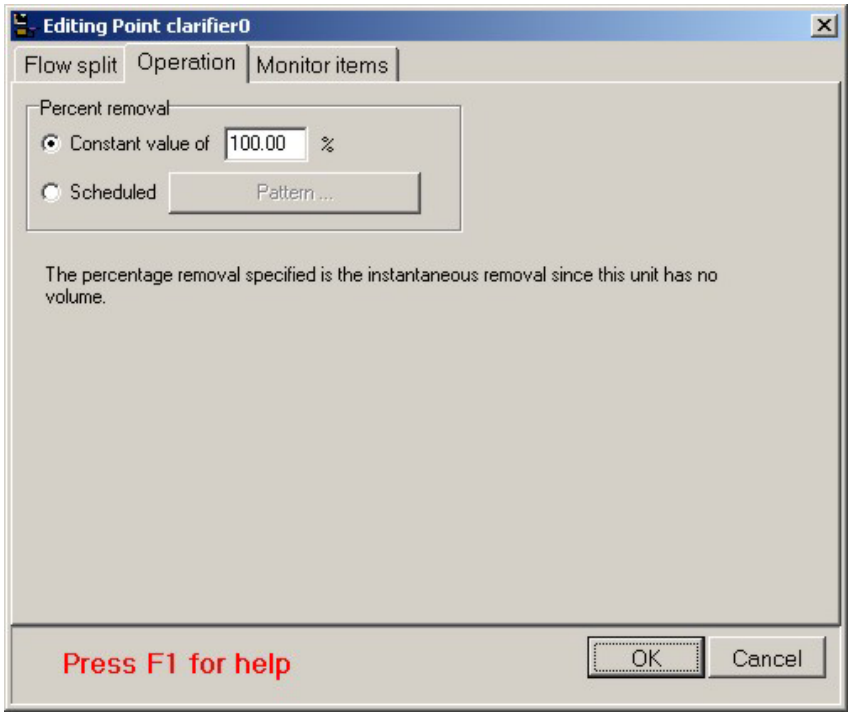
Next to the split method radio group is an edit box; if the split method is selected from the radio group, the value may be entered in the edit box. Units are shown adjacent to the edit box (where applicable) and the label of the edit box changes to indicate the chosen split method.

The split method group also contains a **Constant** check box; if the flow split does not vary with time this box should be checked. It should be noted that this check box applies only to the split methods listed in the radio group and is checked by default. If the **Constant** check box is unchecked, the **Pattern…** button becomes active so you can enter a time-varying pattern for the flow split. Clicking this button presents you with the **Edit split itinerary** dialog box.

The underflow rate also can be **Paced** with an influent stream. To select the flow paced option, click on the **Paced at** check box. The percentage of the influent flow rate may then be specified, and the influent stream for flow pacing may be selected from the drop list box which shows all influent streams available for your system.

Point Secondary Settler Operation

The **Operation** tab, shown below, allows the user to enter configuration specifications for a point secondary settling tank.

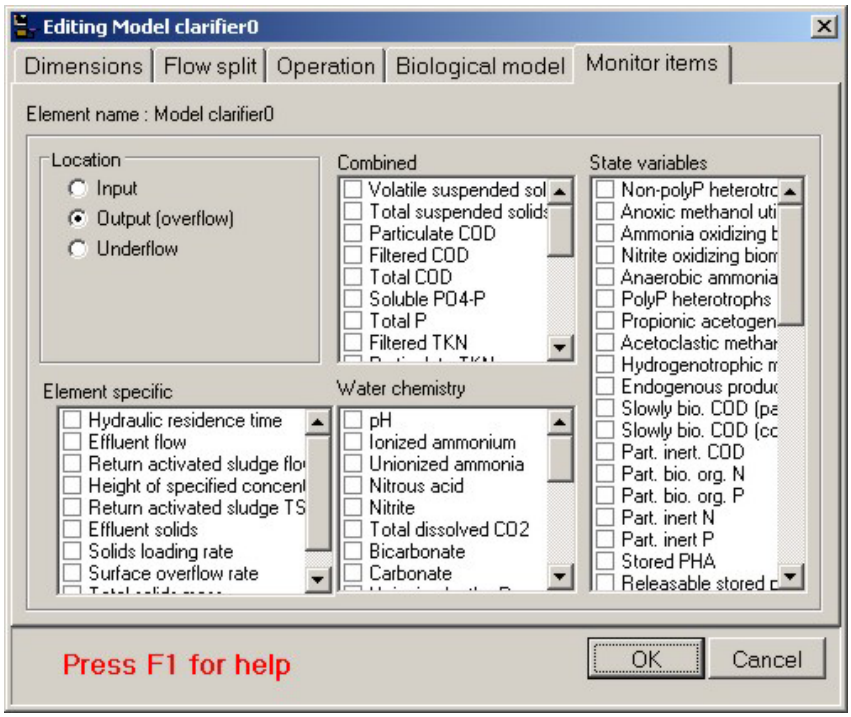


The point secondary settler operation tab

There is an edit box on this tab that is used to enter the **Percent solids removal** for the point secondary settler.

Model Secondary Settler

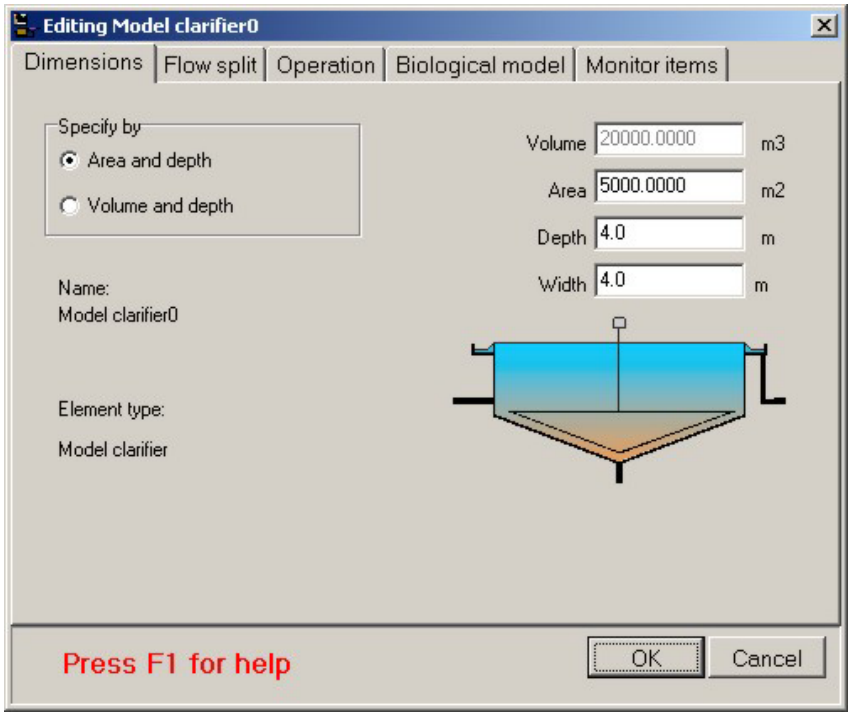
The model secondary settler (clarifier) element is used to model settlement of particulate material in a wastewater stream containing activated sludge mixed liquor based on a one-dimensional flux model. You may specify the physical characteristics, the flow split method, and the solids separation operation of the model secondary settler. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The model secondary settler monitor items tab

Model Secondary Settler Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of a model secondary settler element.

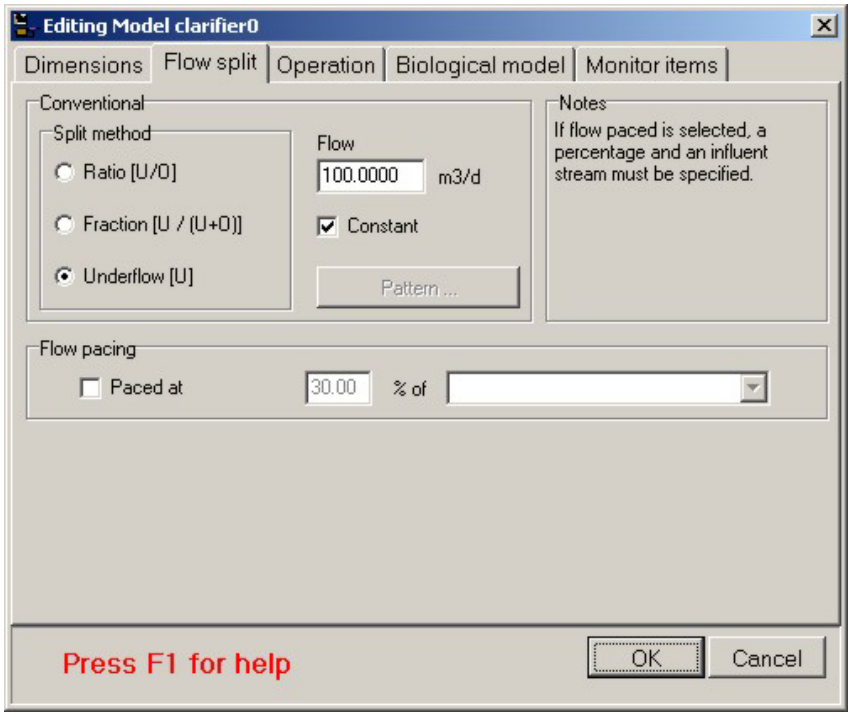


The model secondary settler dimensions tab

There are two methods for entering the dimensions: by **Area and depth**, or by **Volume and depth**. The method is specified by clicking on the appropriate radio button. If you select by area and depth, the model secondary settler area and depth must be entered in the **Area** and **Depth** text edit boxes. If you select by volume and depth, the model secondary settler volume and depth must be entered in the **Volume** and **Depth** text edit boxes. Regardless of the method you choose, you also must specify a **Width** for the element. Units are shown to the right of the edit boxes. The element name and type, and a picture of the element also are shown.

Model Secondary Settler Split Method

The **Flow split** tab, shown below, allows the user to specify the flow split method for a model secondary settler.



The model secondary settler flow split method tab

The method of specifying the flow split for a model secondary settler may be selected from a number of options. You can specify the flow using a **Ratio**, **Fraction**, or **Underflow rate** by clicking on the corresponding radio button. If you specify an underflow rate (denoted by the symbol **U**), it will result in a constant underflow out the bottom of the model secondary settler. Note that when the model secondary settler is operating in this mode, if the influent flow is less than the set **Underflow rate**, then all of the influent flow will be sent to the underflow. If you specify either the **Ratio** or **Fraction** split method, then the underflow will be calculated using the corresponding formula based on the overflow rate (denoted by the symbol **O**) and the underflow rate **U**.

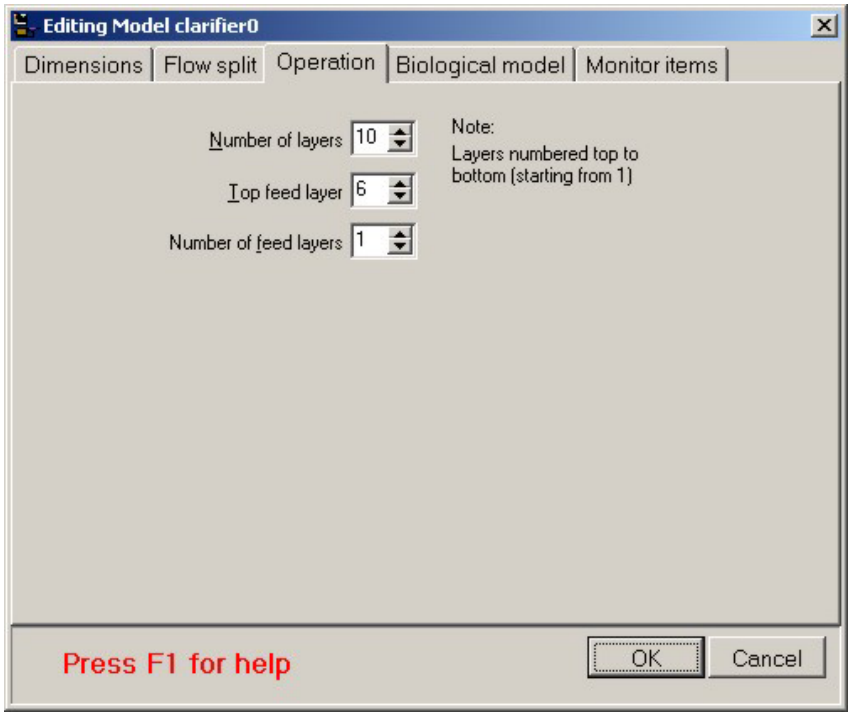
Next to the split method radio group is an edit box; if the split method is selected from the radio group, the value may be entered in the edit box. Units are shown adjacent to the edit box (where applicable) and the label of the edit box changes to indicate the chosen split method.

The split method group also contains a **Constant** check box; if the flow split does not vary with time this box should be checked. It should be noted that this check box applies only to the split methods listed in the radio group and is checked by default. If the **Constant** check box is unchecked, the **Pattern…** button becomes active so you can enter a time-varying pattern for the flow split. Clicking this button presents you with the **Edit split itinerary** dialog box.

The underflow rate also can be **Paced** with an influent stream. To select the flow paced option, click on the **Paced at** check box. The percentage of the influent flow rate may then be specified, and the influent stream for flow pacing may be selected from the drop list box which shows all influent streams available for your system.

Model Secondary Settler Operation

The **Operation** tab, shown below, allows the user to enter configuration specifications for the model secondary settling tank.



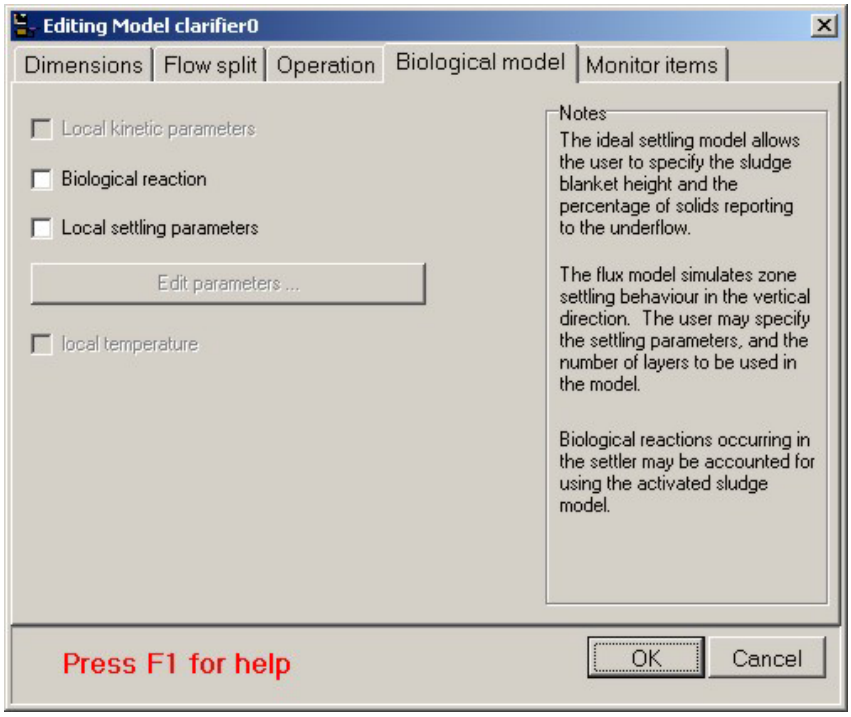
The model secondary settler operation tab

The **Number of layers** used to model the secondary settler can be modified using the spin edit box. The minimum number of layers allowed for the model is 5, the default is 10. Increasing the number of layers will improve the resolution of the predicted settler profile.

You also may specify the **Top feed layer** and the **Number of feed layers** using spin edit boxes. The feed layer refers to the layer into which the applied solids loading is placed. Feed points are not allowed in the top or bottom layers.

Model Secondary Settler Biological Modeling

The model secondary settler **Biological model** tab, shown below, allows the user to specify whether biological reactions in the settler are modeled, and if so, model options such as local kinetic parameters and local temperature.



The model secondary settler biological modeling tab

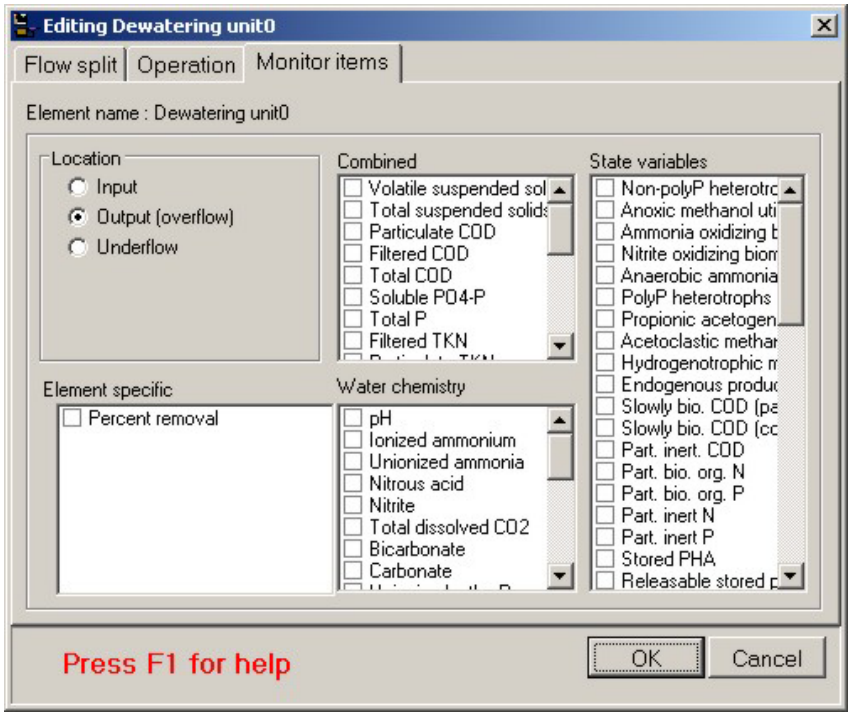
If you wish to model biological reaction in the model secondary settler, click on the check box for **Biological reaction**; this switches on the activated sludge processes and enables the **Local kinetic parameters** and **Local temperature** check boxes.

If you wish to specify **Local kinetic parameters** for the model secondary settler (i.e. kinetic parameters that are different from those specified under the **Project|Model parameters…** menu), click on the check box for local kinetic parameters, then click the **Model parameters…** button to open the **Model parameter editor** dialog box which allows you to access the various activated sludge model kinetic parameters.

A **Local temperature** also may be specified for a model secondary settler. When you click on the check box for local temperature, the **Specify temperature by** radio button group is enabled. You may then specify either a **Constant** or **Scheduled** temperature. If constant temperature is selected, you may enter the value in the edit box. If scheduled temperature is selected the **Pattern…** button becomes active. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

Dewatering Unit

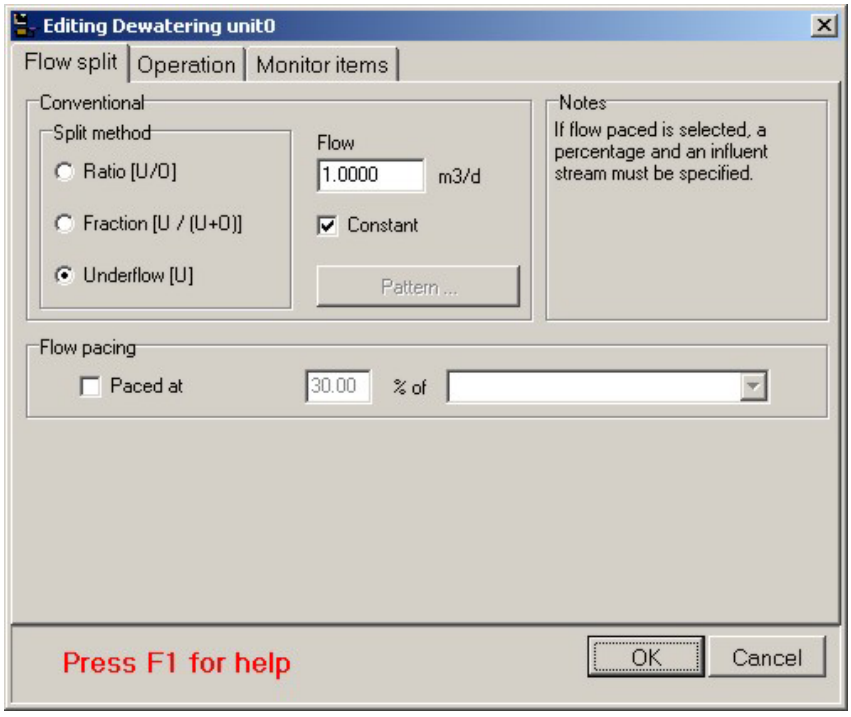
The dewatering unit element used to simulate the separation of liquid and solids in an influent stream to increase the solids concentration. You may specify details regarding the flow split method and the solids separation operation of the dewatering unit. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The dewatering unit monitor items tab

Dewatering Unit Split Method

The **Flow split** tab, shown below, allows the user to specify the flow split method for a dewatering unit.



The dewatering unit flow split method tab

The method of specifying the flow split for a dewatering unit may be selected from a number of options. You can specify the flow using a **Ratio**, **Fraction**, or **Underflow rate** by clicking on the corresponding radio button. If you specify an underflow rate (denoted by the symbol **U**), it will result in a constant underflow out the bottom of the dewatering unit. Note that when the dewatering unit is operating in this mode, if the influent flow is less than the set **Underflow rate**, then all of the influent flow will be sent to the underflow. If you specify either the **Ratio** or **Fraction** split method, then the underflow will be calculated using the corresponding formula based on the overflow rate (denoted by the symbol **O**) and the underflow rate **U**.

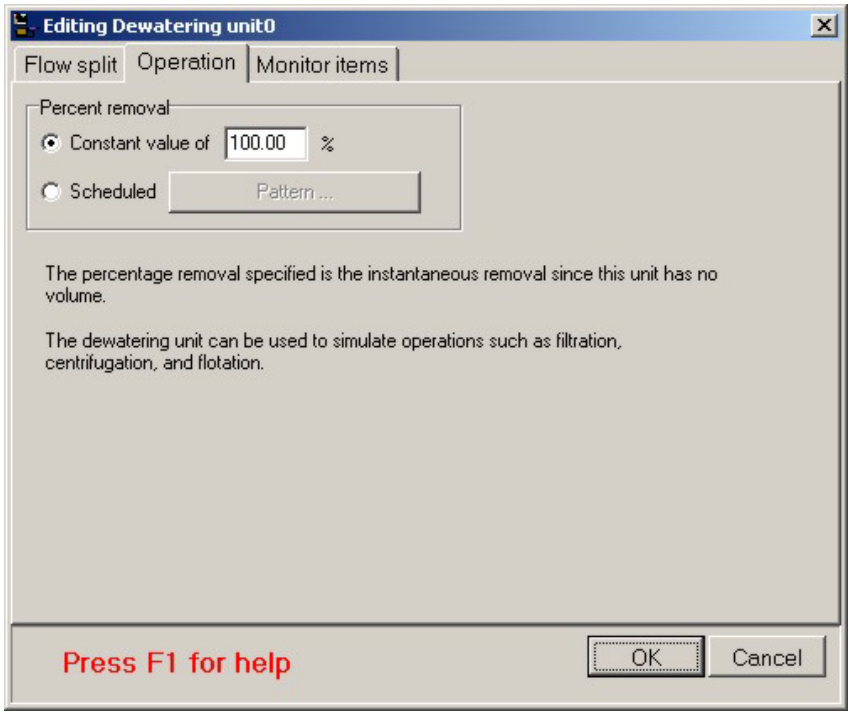
Next to the split method radio group is an edit box; if the split method is selected from the radio group, the value may be entered in the edit box. Units are shown adjacent to the edit box (where applicable) and the label of the edit box changes to indicate the chosen split method.

The split method group also contains a **Constant** check box; if the flow split does not vary with time this box should be checked. It should be noted that this check box applies only to the split methods listed in the radio group and is checked by default. If the **Constant** check box is unchecked, the **Pattern…** button becomes active so you can enter a time-varying pattern for the flow split. Clicking this button presents you with the **Edit split itinerary** dialog box.

The underflow rate also can be **Paced** with an influent stream. To select the flow paced option, click on the **Paced at** check box. The percentage of the influent flow rate may then be specified, and the influent stream for flow pacing may be selected from the drop list box which shows all influent streams available for your system.

Dewatering Unit Operation

The **Operation** tab, shown below, allows you to specify the degree of thickening obtained by the dewatering unit.

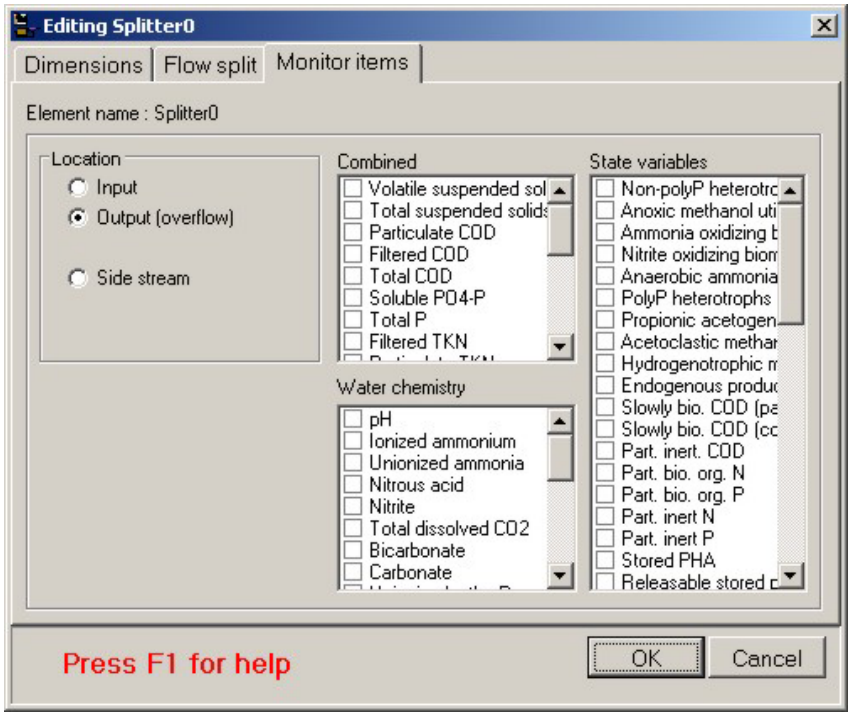


The dewatering unit operation tab

The edit box on this tab is used to enter the **Percent solids removal**.

Splitter

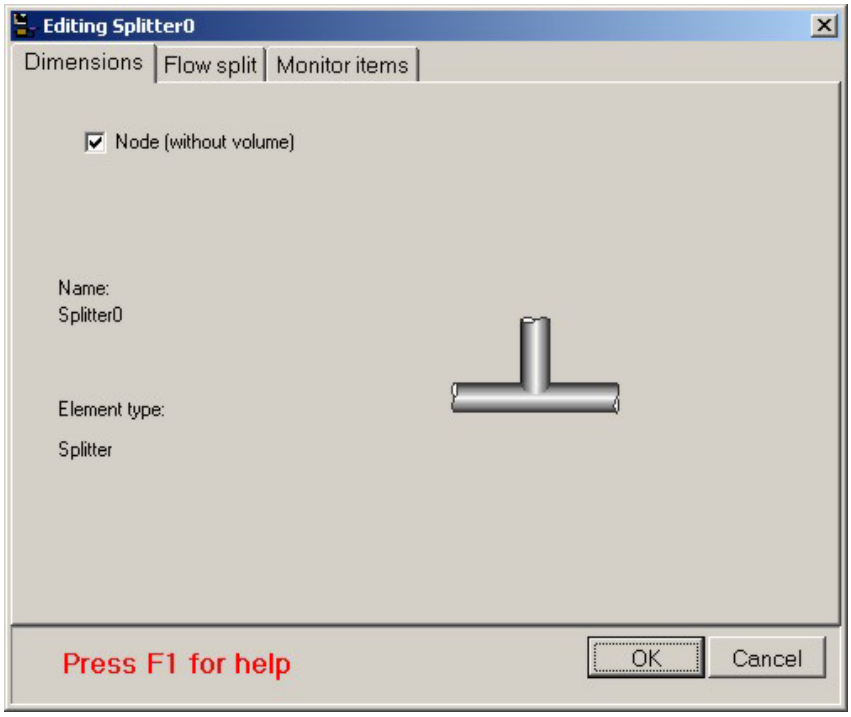
Splitters divide a stream on a mass / flow (constant density) basis. You may specify physical characteristics of the splitter and the method used by the splitter to split the influent flow. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The splitter monitor items tab

Splitter Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of a splitter element.

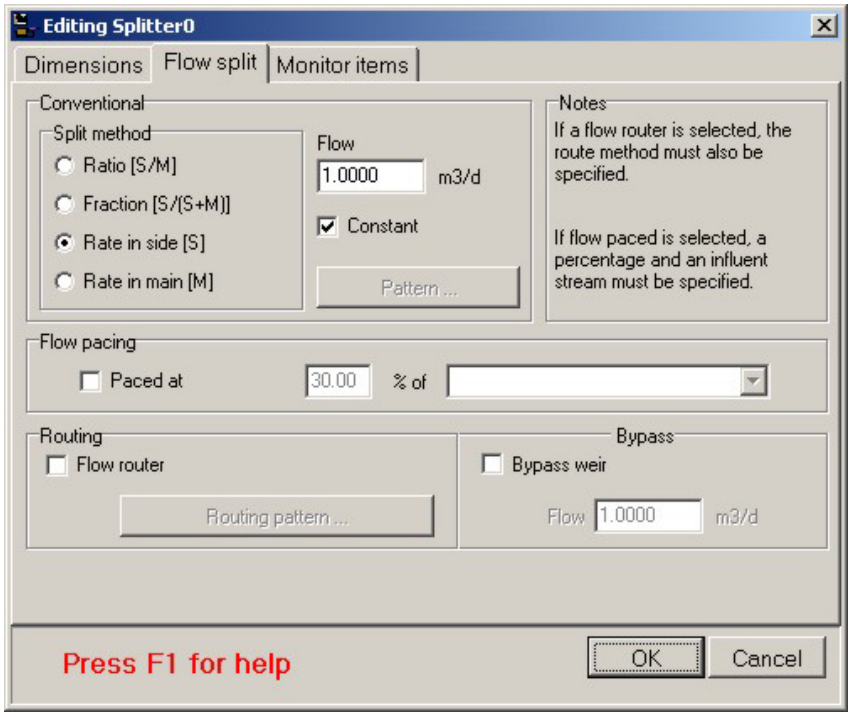


The splitter dimensions tab

Flow splitters are assumed to be dimensionless unless otherwise specified. The dimensions tab for elements which do not require a volume such as splitters contains a **Node (without volume)** check box; if you wish to enter a volume, deselect this option. You may then enter the splitter **Volume** in the edit box. Note that if you are using volumeless (i.e. node) splitters in your configuration you should be aware of recursion in your flow network. Units are shown to the right of the edit box. This tab also shows the element name and type, and a picture of the element.

Splitter Flow Split

The **Flow split** tab, shown below, allows the user to specify the type of splitter and the flow specifications.



The splitter flow split method tab

There are four different splitter types, each of which may be configured from this tab. These are:

1. Conventional
2. Flow paced
3. Router
4. Bypass

A conventional splitter can operated on a constant or pattern basis. To operate in the constant mode, select the box labeled **Constant**. Next you must select a method used to split the flow using the **Specify split method** radio button group (**Ratio**, **Fraction**, **Rate in side,** or **Rate in main**). Once you have chosen a split method, you must enter the value for the ratio, fraction, or flow in the text edit area that will be labeled according to the choice you have made. If you specify a maximum rate in the side or main output stream (denoted by the symbols **S** and **M**, respectively), it will result in a constant flow to the stream. Note that when these methods are used, if the influent flow is less than the specified rate, then all of the influent flow will be sent to the stream for which the rate was specified. If you specify either the **Ratio** or **Fraction** split method, then the output stream flows will be calculated using the corresponding formula based on the side and main output stream flows.

If the **Constant** check box is unchecked, the **Pattern…** button becomes active so you can enter a time-varying pattern for the flow split. Use the **Specify split method** radio button group (**Ratio**, **Fraction**, **Rate in side,** or **Rate in main**) to choose the split method that you want to set a pattern for. Next, click the **Pattern…** button to access the **Edit split itinerary** dialog box which you may use to enter your pattern.

Options for a flow paced splitter are found in the **Flow pacing** group. Select the **Paced at** check box. You may now specify the remaining two items required for a

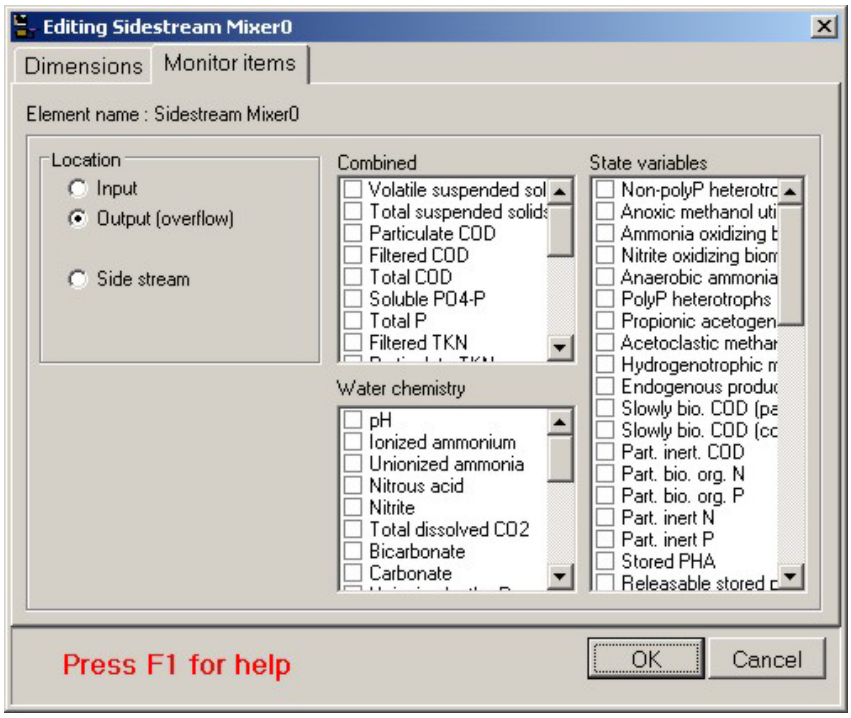
flow paced splitter. Enter a number in the text edit area labeled **% of**, and then select an influent element as the basis for flow pacing from the drop list box of influent elements in the configuration. Note that when you select to use a flow paced splitter, the side stream output is paced as a percentage of the selected influent.

Options for a router are located in the **Routing** group. Select the **Flow router** check box. Click on the **Routing pattern…** button which will now be enabled to open the **Edit router itinerary** dialog box. Note that when you select to use a router, you are using a timed pattern to switch between a fraction of 0 and 1. That is, all flow goes either to the main or side output stream according to the time interval pattern you specify.

Options for a bypass splitter are located in the **Bypass** group. Select the **Bypass weir** check box. When you do this, you will be able to enter a value in the **Flow** text edit area. This value will set the maximum allowable flow in the main output stream. Any flow in excess of this value will bypass the main output stream and be directed to the side output stream.

Side Stream Mixer

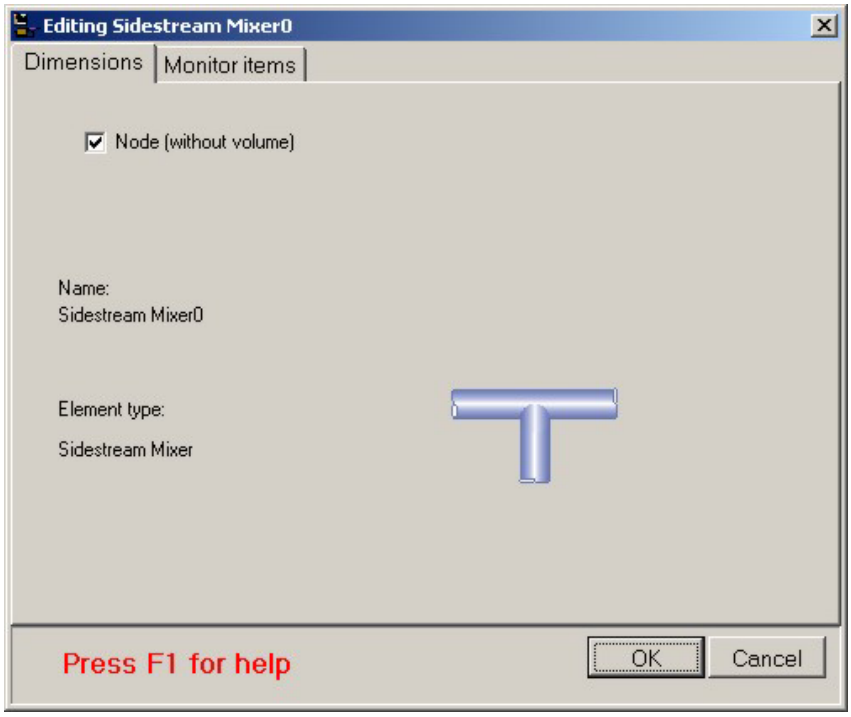
Mixers combine multiple streams into a single stream. You may specify details regarding the physical characteristics of the mixer element. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The mixer monitor items tab

Side Stream Mixer Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of a mixer element.



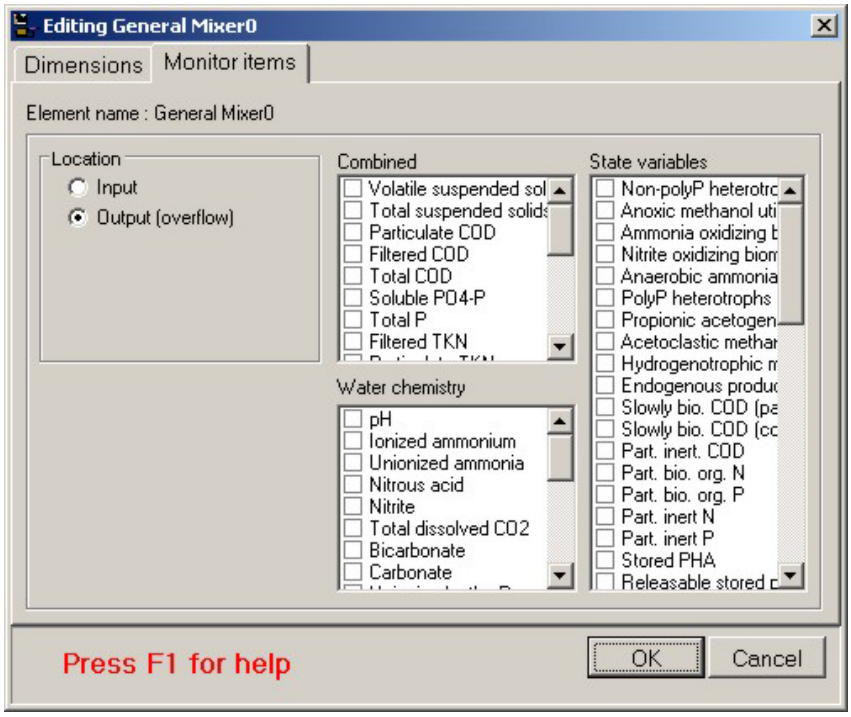
The mixer dimensions tab

Mixers are assumed to be dimensionless unless otherwise specified. The dimensions tab for elements which do not require a volume such as mixers contains a **Node (without volume)** check box; if you wish to enter a volume, deselect this option. You may then enter the mixer **Volume** in the edit box. Note that if you are using volumeless (i.e. node) mixers in your configuration, you should be aware of recursion in your flow network. Units are shown to the right of the edit box. This tab also shows the element name and type, and a picture of the element.

Generally mixer volumes are very small in comparison with other vessels in a configuration. Consequently these elements often may be modeled as nodes without volume. The use of node elements in BioWin will substantially improve simulator performance for both dynamic and steady state simulations. The mixer with volume is modeled as a "completely mixed" sump.

General Mixer

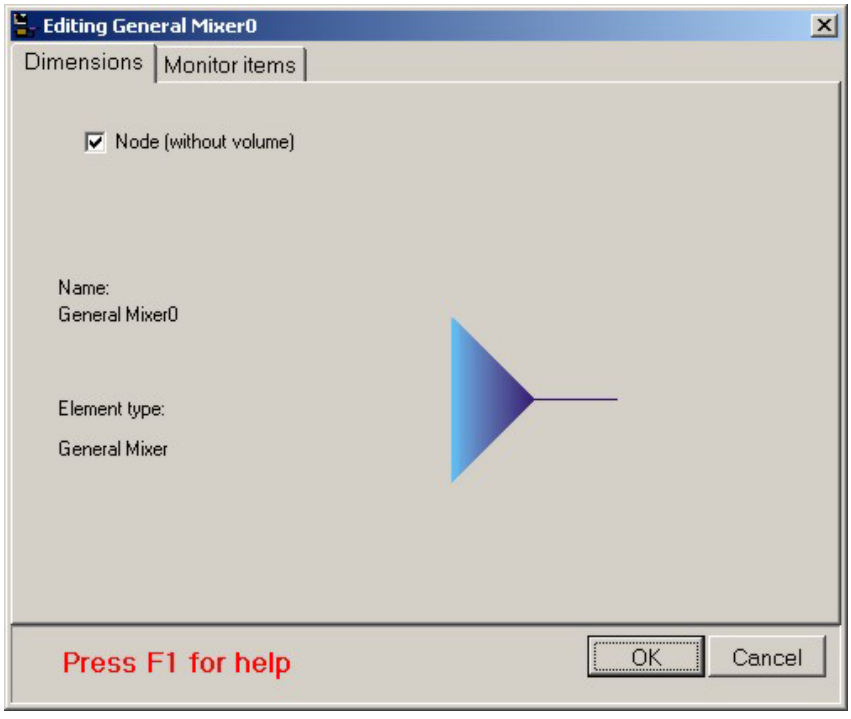
Mixers combine multiple streams into a single stream. You may specify details regarding the physical characteristics of the mixer element. For information on monitoring parameters/variables for this element, please see **Monitoring Data**.



The general mixer monitor items tab

General Mixer Dimensions

The **Dimensions** tab, shown below, allows the user to enter the physical dimensions of a mixer element.



The mixer dimensions tab

Mixers are assumed to be dimensionless unless otherwise specified. The dimensions tab for elements which do not require a volume such as mixers contains a **Node (without volume)** check box; if you wish to enter a volume, deselect this option. You may then enter the mixer **Volume** in the edit box. Note that if you are using volumeless (i.e. node) mixers in your configuration, you should be aware of recursion in your flow network. Units are shown to the right of the edit box. This tab also shows the element name and type, and a picture of the element.

Generally mixer volumes are very small in comparison with other vessels in a configuration. Consequently these elements often may be modeled as nodes without volume. The use of node elements in BioWin will substantially improve simulator performance for both dynamic and steady state simulations. The mixer with volume is modeled as a "completely mixed" sump.

BioWin Album

Parts of the Album

The BioWin album provides users with a powerful tool for displaying results of simulations. It provides you with a means for displaying simulation results in a variety of manners. You may display your simulation results in a numerical format using BioWin's customizable table and element specific information displays.

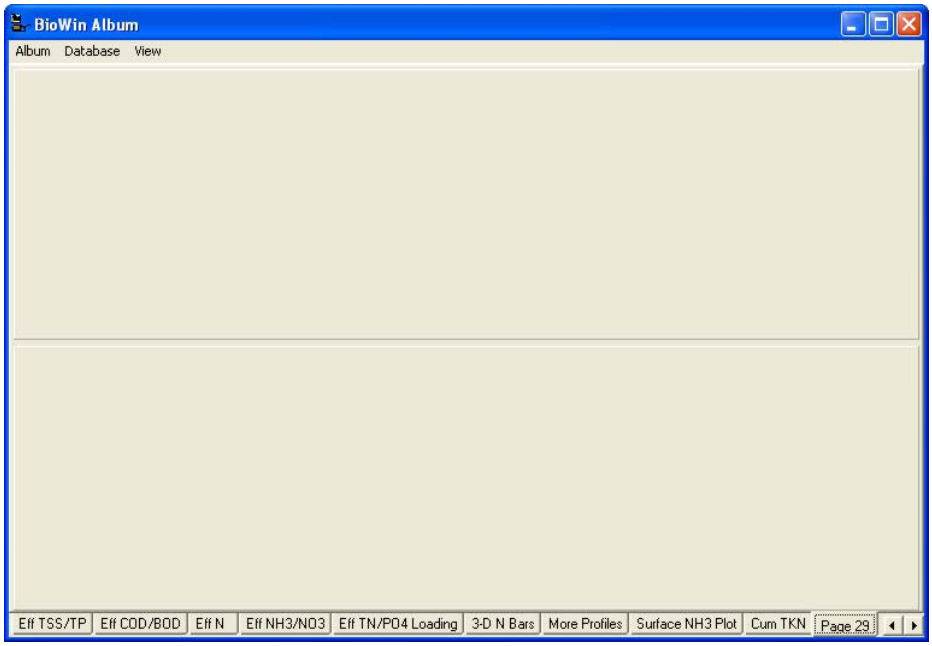
Perhaps the greatest strength of the album is that it provides you with an interface for displaying your simulation results in a variety of rich graphical formats. Because these charting capabilities are so powerful, there are three major sections of the manual devoted to them. For information related to creating charts, please see "*Creating Charts & Adding Series*". For information on chart formatting, please see *Chart Formatting Procedures*". Finally, "*Series Formatting Procedures*" contains information detailing the various formatting options for different series styles.

The BioWin album is accessed from the main simulator window via the menu choice **View|Album**. Note that once you have set up the album to appear as you desire, closing it *will not* result in a loss of your changes. The album and the information it contains are saved along with the BioWin configuration file.

The BioWin album consists of three main parts:

1. Menus
2. Pages
3. Panes

The following sections give an outline of each part.



The BioWin album

Album Menus

The various menus available in the BioWin album are located at the top of the window. The menus available are:

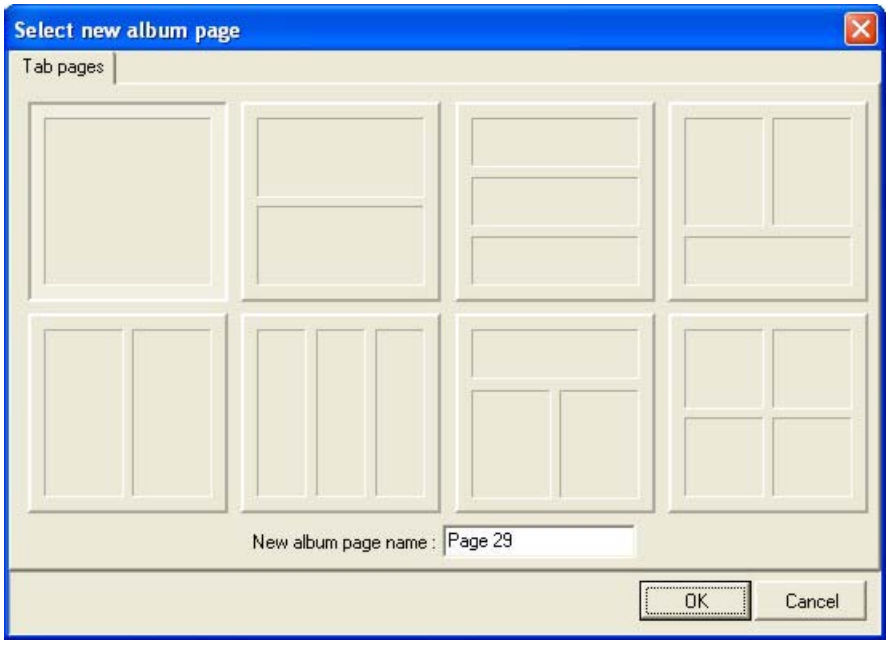
- **Album**
- **Database**
- **View**

Each menu may be accessed with either of two methods:

1. Click on the text of the menu, or;
2. Hold down the **Alt** key on your keyboard and press the letter on the keyboard corresponding to the underlined letter in the menu title. For example, **Alt+A** will access the **Album** menu.

Album Pages and Panes

Pages are used as "containers" for *panes* in the BioWin album. You may have as many pages as you desire in the album. Only one page at a time may be viewed; you may select a page for viewing by clicking on its tab. Each page can contain up to four panes, and BioWin offers a number of different pane layouts on any page as shown in the picture below.



BioWin album page layout gallery

In most cases, the panes on a page are resizable to offer further flexibility. Panes are used as "containers" for the various information displays in the BioWin album, i.e.:

1. Charts
2. Element information
3. Tables

After you have assigned one of these displays to a pane, it is possible to switch to one of the other remaining two.

Since there is a large amount of information related to charts, they will not be discussed in this chapter. . For information related to charts, please see "*Creating Charts & Adding Series*", *Chart Formatting Procedures*" and "*Series Formatting Procedures*". For information regarding (2) and (3), see the relevant sections later in this chapter. Next, a number of common procedures related to album pages and panes will be covered.

Add a Page to the Album

1. Click **Album|Add Page…** (you also can access this command by right-clicking on an existing album page's tab).
2. Click on the pane layout for the page from the gallery of available choices.
3. If you want a page name different from the default given, enter a name for the new page in the **New album page name:** text edit area. This is the text that appears in the page tab.
4. Click **OK** to finish adding the page.

Move a Page in the Album

1. Right-click the tab of the album page that you wish to move (i.e. change order of).

2. Select **Move page** from the resulting popup menu and you will be presented with the dialog box shown below.

Dialog box used to move pages around in the album

3. The dialog box will list the other pages currently contained in the album. Select the page that you want to place the page you are moving in front of, and click **OK**.

Note: If you double-click on a page name listed in the **Move page** dialog box, the page you are moving will be placed in front of that page *before* you click **OK**. You can use this feature to quickly "try out" several locations for the page you are moving.

Delete a Page from the Album

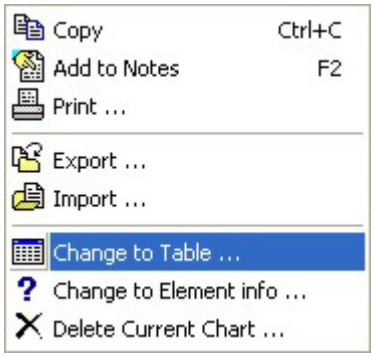
1. Click on the tab of the album page you want to delete to make it the current page.
2. Select **Album|Delete current page** (you also may access this command by right-clicking on the tab of the page you wish to delete).

Rename An Album Page

1. Click on the tab of the album page you wish to rename to make it the current page.
2. Select **Album|Rename current page** (you also can access this command by right-clicking on the tab of the page you wish to rename).

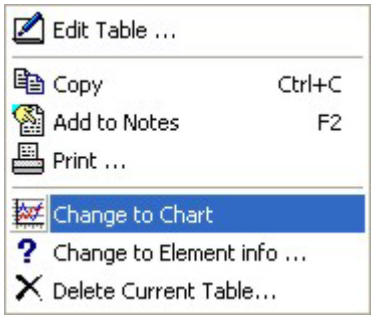
Switch Display Types

1. Right-click on the album chart, table, or element information display that you wish to replace with another type.
2. If you right-click on a chart and select the **Chart** option you will be presented with the dialog box shown below.



Dialog Box used to switch from chart to table or element information display

3. From the sub-menu, select one of the possible two other types.
4. If you right-click on a table or element information display you will be presented with the dialog box shown below.



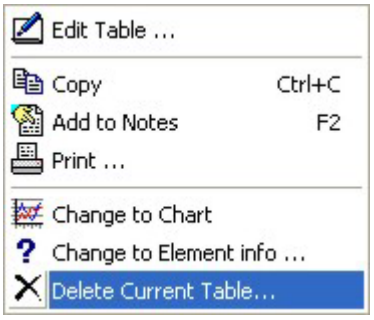
Dialog Box used to switch to chart or element information display

5. From the sub-menu, select one of the possible two other types.

Delete a Display

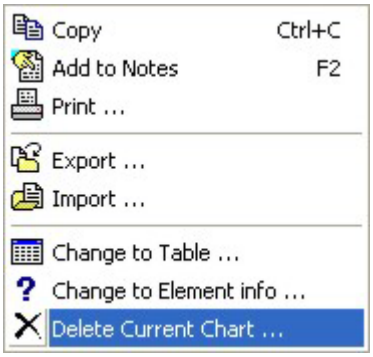
Right-click on the album chart, table, or element information display that you wish to delete.

- If you are deleting a table or element information display, select the **Delete Current...** option from the resulting popup menu as shown below.



Dialog Box used to delete current table

- If you are deleting a chart, select the **Chart** option from the resulting popup, then select **Delete Current Chart…**from the resulting flyout.



Dialog Box used to delete current chart

Resize Panes

Hold the mouse cursor over a dividing line adjacent to the album pane that you wish to resize.

Horizontal Resize

Vertical Resize

- If the dividing line is vertical, your cursor will change to the horizontal resize cursor. Click the mouse button and drag the dividing line to change to horizontal size of the pane.
- If the dividing line is horizontal, your cursor will change to the vertical resize cursor. Click the mouse button and drag the dividing line to change the vertical size of the pane.

Note: The panes in the layout that consists of four equally sized panes on a page may not be resized. This is the only layout where resizing of panes is not permitted.

Album Page Templates

If you have set up a set of album pages that you would like to re-use as a template in another BioWin project, you can save them as Album Page Template files (*.atp). For more information, please see the following section.

Opening Album Pages

The album **File|Open pages** command will present you with the **Open** dialog box set to look for files with the extension **.atp* (album template pages).

Use this dialog box to locate the **.atp* pages that you want to open. When you have located the file, you may open it by double-clicking on it, or clicking it and then clicking **Open**. When you open the **.atp* file, the pages and their associated panes and information displays (i.e. charts, tables, element) will be added to the BioWin album. Note that any pages already in the album *are not* overwritten. Also, if any of the charts in the **.atp* file contain series, those series will not be contained in the

current database; essentially they are inactive lines drawn on the chart. If you want to add series to the charts that you have added via this command, you may do so.

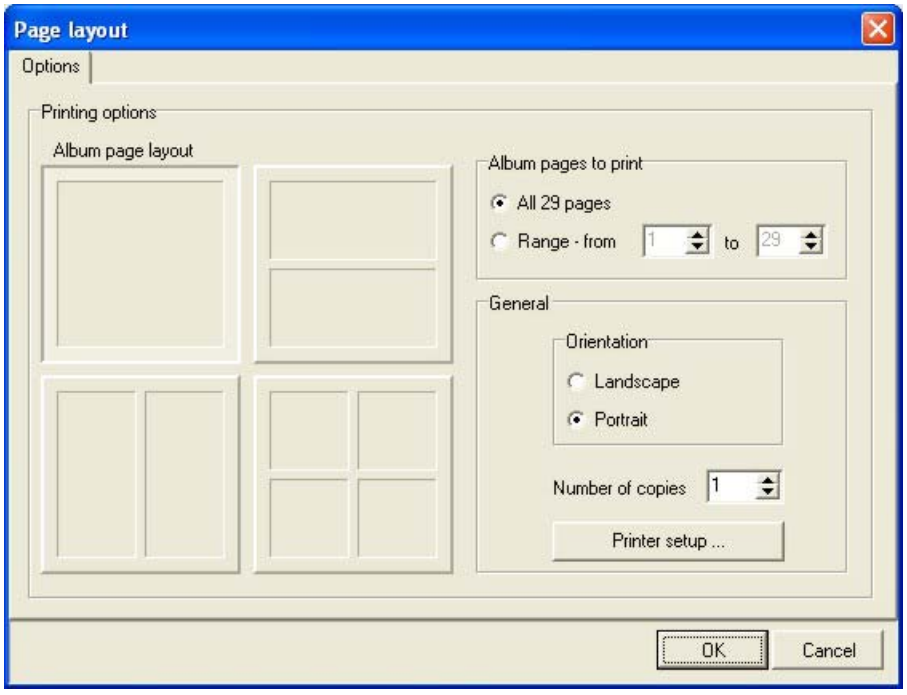
Save Album Pages

The album **File|Save pages** command saves the current group of album pages and allows you to specify a name and location. When you choose this command, you will be presented with the **File Save** dialog box. When you have done this, type the name you want for the file in the **File name:** text edit area. Note that you do not have to specify the file extension – BioWin will add the proper extension (*.atp*) to your file. When you are satisfied with your file name, click the **Save** button. When you save a file, the following information is saved:

- The pages, their names, and location in the workbook relative to each other;
- The pane layout on each page;
- The type of information display on each pane (i.e. chart, table, element summary);
- Formatting of charts and series with their data. Note that when you save a chart in this manner, *only* the formatting is saved – series links to the database will not be saved and the series essentially become inactive lines.

Printing Album Pages

The **Album|Print pages** command should be used when you want to print multiple or all of the album pages. Selecting this command will present you with the dialog box shown below.



Dialog box used for printing multiple or all of the album pages

The **Album pages to print** section allows you to choose between printing **All X pages** (X changes according to the number of pages in the current album), or a **Range** of pages (when the option is selected, you may enter text in the **from** and **to** spin edit boxes).

Once you have selected the album pages that you want to print, you can choose between one of four **Album page layout** options:

1. The top left option will result in the contents of each album page being printed on an entire page. For example if an album page contains only one chart, that chart will be printed on an entire page. If an album page contains two panes with a chart on each, then the two charts will be printed on an entire page.
2. The top right option will print the contents of each album page to one horizontal half of a page. Each printed page will contain two BioWin album pages.
3. The bottom left option will print the contents of each album page to one vertical half of a page in the report. Each printed page will contain two BioWin album pages.
4. The bottom right option will print the contents of each album page to one quarter of a page in the report. Each printed page will contain four BioWin album pages.

The **General** section contains a number of generic printing options. You can select your page **Orientation** to be **Landscape** or **Portrait**. You can specify the **Number of copies** of each page that you would like to print. The **Printer setup…** button will open the printer setup dialog box which will allow you to access the printer's properties, set paper size, page orientation, and a number of other printer options (the options presented to you will be dependant on the printer you have).

Once you are satisfied with your layout settings, click the **OK** button to print.

Album Element Information Displays

BioWin has two preset element information displays for viewing in the album. They provide a quick means of displaying information about a particular element in your configuration. Like any other information display, these may be added to any pane on a page.

Element specific information includes parameters that are relevant only to certain types of elements. For example, Solids Loading Rate only applies to settlers, Oxygen Utilization Rate applies only to aerated bioreactors, etc.

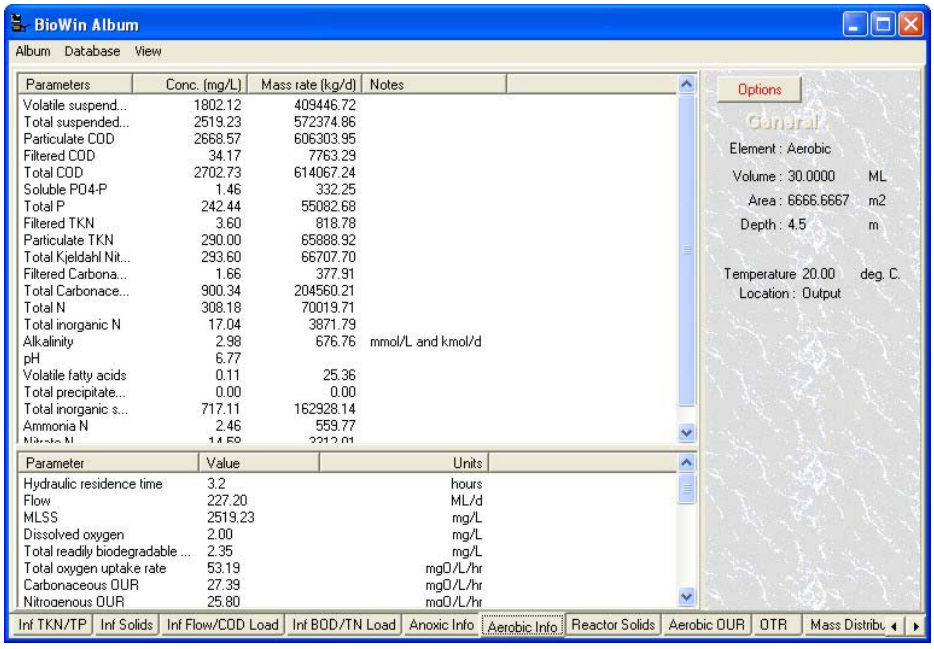
The two types of element information display are **State variable** and **Summary**. Both types of display are divided into two main sections. The **State variable** display lists information about all the state variables for the selected element, including their concentrations and mass flow rates in the left section. The **Summary** display lists information about various compounds (e.g. Volatile Suspended Solids, Chemical Oxygen Demand, etc.) for the selected element, including their concentrations and mass flow rates in the left section. In the bottom section of the display, element specific information is displayed.

In the right section, both types of display show general information about the selected element such as the element name, the location in the element from where the compound / state variable data are obtained (i.e. Input, Output, Underflow), and where appropriate, physical data such as element dimensions and temperature. You may control the location in the element from where the compound / state variable data are obtained by clicking the **Options** button. Note that this may not be

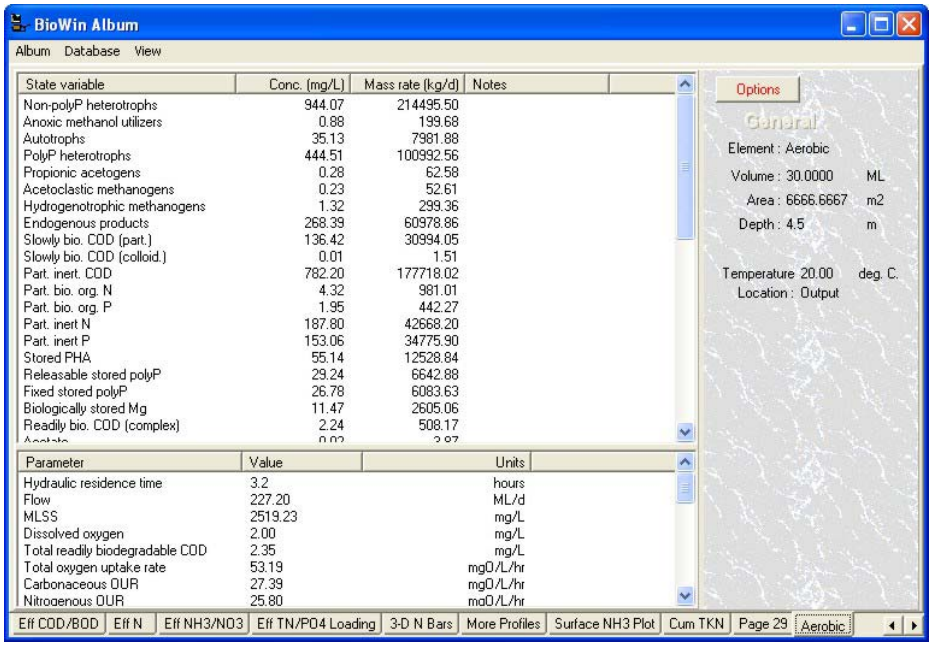
appropriate for all element types since for certain elements (e.g. effluent) the location is irrelevant. For splitters, if you want to see data for the side stream, you should select the **Alt. output** option.

Both types of display also allow you to resize the columns in the left section where names, concentrations, and mass flow rates of state variables / compounds are listed. Also, you can change the space allocated in the pane for each section by holding your mouse over the vertical or horizontal line which divides the two sections of the display until the resize cursor appears, and dragging the vertical line.

Example element information displays are shown below.

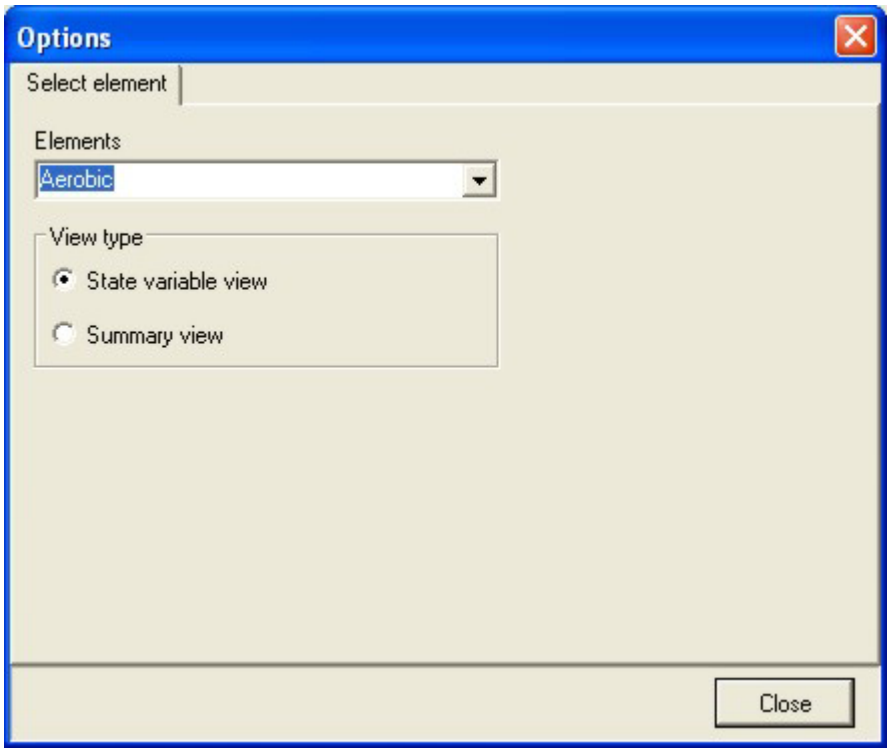


A "summary" element information display



A "state variable" element information display

Add an Element Info Display From The Album



Tab used for creation of element information displays

1. Right-click on the album pane where you want to place an element information display.
2. Select **Element info** from the resulting pop-up menu.
3. In the **Elements** drop list box, select the element that you want to display information for.
4. In the **View type** radio button group, select **State variable view** or **Summary view**.
5. Click **Close** to finish.

Add an Element Info Display From the Drawing Board

 *Element Selection Tool*

 *Selection Cursor*

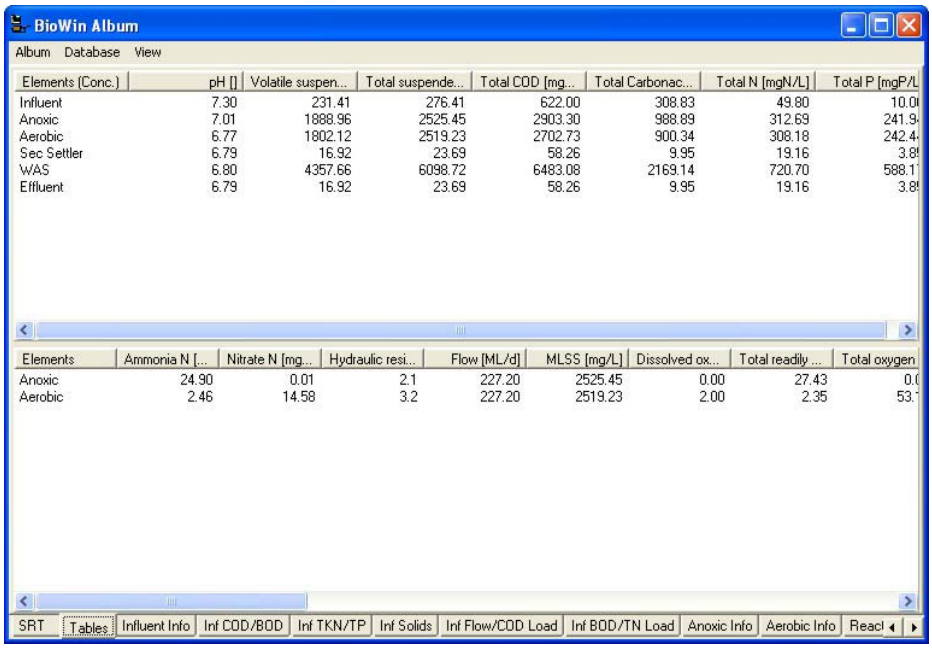
1. Click on the element selection tool from the configure toolbar.
2. Move the cursor over the element on the drawing board for which you wish to add an element information display. When you do so, the cursor will change to the element selection cursor.
3. Right-click on the element, and select Add to album from the resulting popup menu.

4. From the resulting flyout menu, select either Element info (Summary) or Element info (State variables). A new page will be added to the album for the element information display you have selected.

Album Table Displays

To allow users more flexibility for displaying information about elements, BioWin also offers the functionality of creating customized tables for display in the album. In these tables, it is possible to include all or any combination of the elements in the current configuration. You also may choose the compounds and/or variables included in the table. Once you have added a table, you may add more elements to it or remove existing ones if you wish. You also can resize the columns in the table.

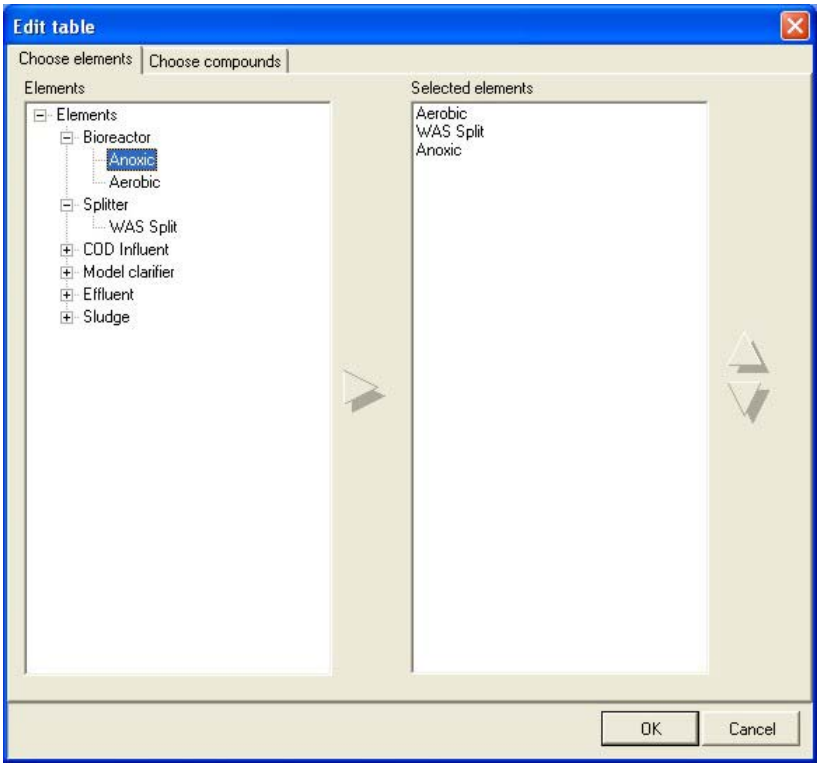
A two-pane album page with example tables is shown below.



An album page showing two example tables

Add a Table Display

1. Right-click on the blank album pane where you wish to place the table.
2. Select **Table** from the resulting pop-up menu.



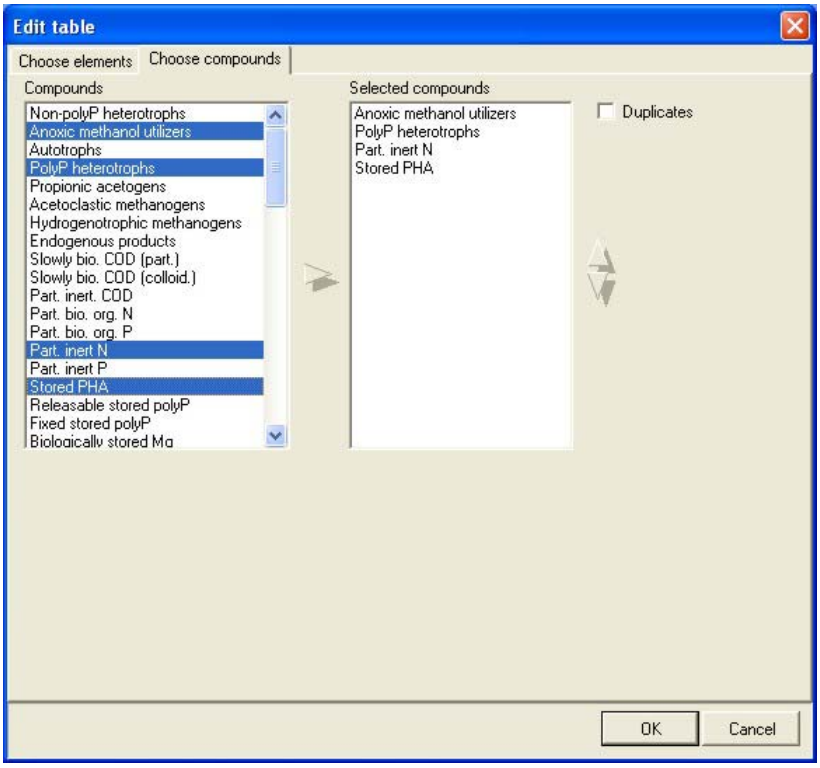
Tab used to choose elements for table display

3. A dialog will open with the **Choose elements** tab displayed. From the **Elements** tree view, select the element(s) that you wish to include in the table.

   - If you wish to include all the elements from a group (for example, all the Bioreactors), click on the group heading and then click the right-pointing arrow to move them all to the **Selected Elements** list.
   - If you wish to include only certain elements from a group (or groups), then click on the plus sign (+) next to the group heading to expand it, click on the specific element you want to include, and then click the right-pointing arrow to move it to the **Selected Elements** list.
   - If you want to change the order in which the **Selected Elements** will appear in the table, move the elements around by clicking on them and then clicking the Up/Down arrows.

4. Now click the **Choose compounds** tab. In the **Compounds** list, select the compounds that you wish to appear in your table, and add them to the **Selected compounds** list by clicking the right-pointing arrow. To select multiple compounds;

   - Select contiguous multiple compounds by clicking on the first desired compound, holding down the **Shift** key, and clicking the last desired compound.
   - Select non-contiguous multiple series by holding the **Ctrl** key and clicking on the different desired compounds.



Tab used to choose compounds for inclusion in table display

5. If you wish to re-add certain compounds, place a check in the box labeled **Duplicates**, and re-add the compounds.
6. If you want to change the order in which the compounds will appear in the table, move the compounds around by clicking on them and then clicking the Up/Down arrows.
7. Click **OK** to finish.

Edit Table Entries

To add to the elements listed in the table:

1. In the album, double-click on any row of the table, or right-click and select **Edit Table…** from the resulting popup menu.
2. Click the **Choose elements** tab of the resulting dialog box.
3. If you wish to include all the elements from a group (for example, all the Bioreactors), click on the group heading and then click the right-pointing arrow to move them all to the **Selected Elements** list.
4. If you wish to include only certain elements from a group (or groups), then click on the plus sign (+) next to the group heading to expand it, click on the specific element you want to include, and then click the right-pointing arrow to move it to the **Selected Elements** list.
5. Click **OK** to finish.

To delete an element from the table:

1. Double-click on any row of the table, or right-click and select **Edit Table…** from the resulting popup menu.
2. Click the **Choose elements** tab of the resulting dialog box.
3. In the **Selected elements** list, click on the element you wish to remove and press the **Delete** key on your keyboard.
4. Click **OK** to finish.

To change the order of the elements in the table:

1. Double-click on any row of the table, or right-click and select **Edit Table…** from the resulting popup menu.
2. Click the **Choose elements** tab of the resulting dialog box.
3. Change the order of the elements in the **Selected elements** list by clicking on elements and clicking the Up/Down arrows.
4. Click **OK** to finish.

To add compounds to the table:

1. Double-click on any row of the table, or right-click and select **Edit Table…** from the resulting popup menu.
2. Click the **Choose compounds** tab of the resulting dialog box.
3. In the **Compounds** list, select the compounds that you wish to appear in your table, and add them to the **Selected compounds** list by clicking the right-pointing arrow. Select contiguous multiple compounds by clicking on the first desired compound, holding down the **Shift** key, and clicking the last desired compound. Select non-contiguous multiple compounds by holding the **Ctrl** key and clicking on the different desired compounds.
4. If you wish to re-add certain compounds, place a check in the box labeled **Duplicates**, and re-add the compounds.
5. Click **OK** to finish.

To delete a compound from the table:

1. Double-click on any row of the table, or right-click and select **Edit Table…** from the resulting popup menu.
2. Click the **Choose compounds** tab of the resulting dialog box.
3. In the **Selected compounds** list, click on the compound you wish to remove and press the **Delete** key on your keyboard.
4. Click **OK** to finish.

To change the order of the compounds in the table:

1. Double-click on any row of the table, or right-click and select **Edit Table…** from the resulting popup menu.

2. Click the **Choose compounds** tab of the resulting dialog box.
3. Change the order of the compounds in the **Selected compounds** list by clicking on compounds and clicking the Up/Down arrows.
4. Click **OK** to finish.

Resize Table Columns

You can resize columns in a table in one of two ways:

Horizontal Resize

- Open the album page containing the table you want to resize columns for. Hold your cursor over the right dividing line of the column you want to resize in the column heading row. When you do this, your cursor will change to the horizontal resize cursor. Click the mouse and drag the column dividing line until the column is the desired size.
- To size a column so that it fits the widest value displayed in that column, simply click on the column heading. Note that this will undo any resizing you have done with the first method as it sets the other columns to a "standard" width.

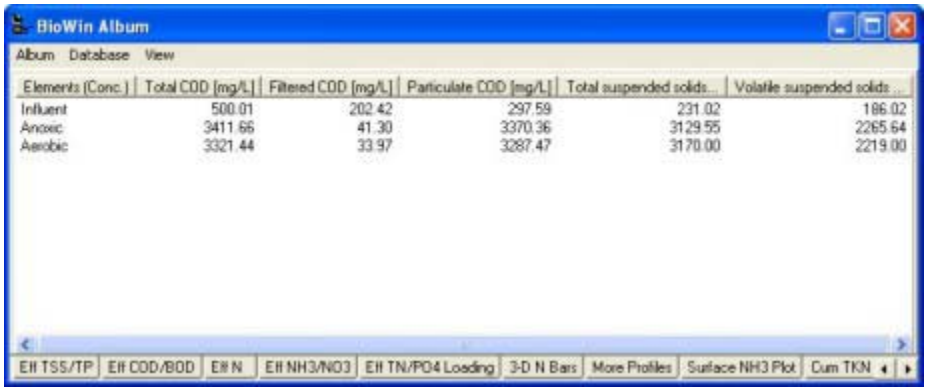
Specialized Types of Table Displays

There are a number of tables that display data that are different from or not available in standard tables. A summary of these specialized tables is given in the following sections. Specialized tables are added to the album and edited similarly to standard tables, as outlined in the previous sections. Any procedures that differ from the standards are highlighted below.

Mass Rate Tables

To allow users more flexibility for displaying information about elements, BioWin offers the functionality of creating customized mass rate tables for display in the album. In these tables, it is possible to include all or any combination of the elements in the current configuration. You also may choose the compounds and/or variables included in the mass rate table. Once you have added a mass rate table, you may add more elements to it or remove existing ones if you wish. You also can resize the columns in the table. The mass rates shown in these tables are in kg/d if Metric flow units are being used, and lb/d if Imperial flow units are being used.

An album page with an example mass rate table is shown below.



An album page showing a mass rate table

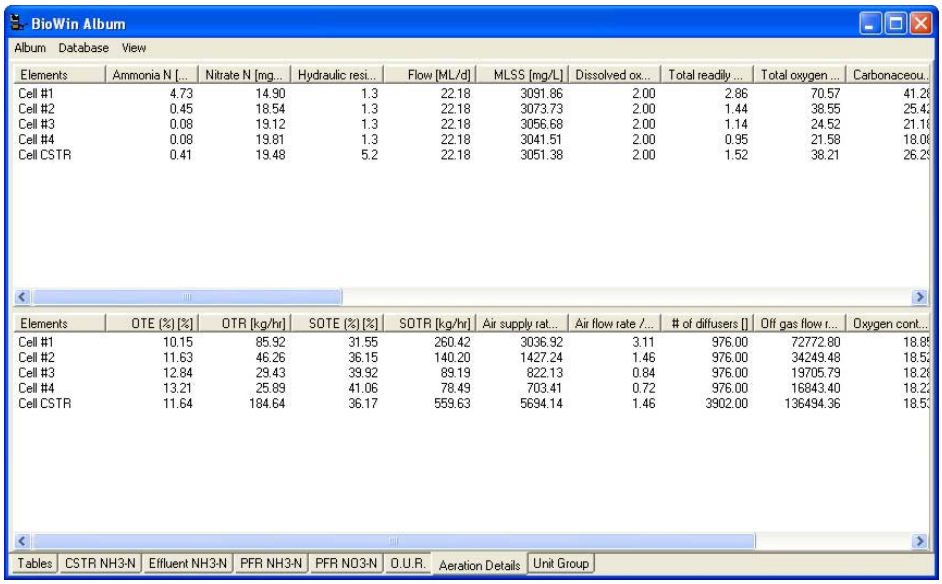
Add a Mass Rate Table

1. Right-click on the album pane where you wish to place the mass rate table.
2. Select **Mass rate table…** from the resulting pop-up menu.
3. Continue as you would for a standard table.

Aeration tables

To allow users more flexibility for displaying information about bioreactor-class elements BioWin also offers the functionality of creating tables especially for the display of aeration details in the album. In these tables, it is possible to include all or any combination of the bioreactor-class elements in the current configuration. You also may choose the compounds and/or variables included in the aeration table. Once you have added an aeration table, you may add more bioreactor-class elements to it or remove existing ones if you wish.

A two-pane album page with example aeration tables is shown below.



Example aeration tables (top pane shows a default aeration table, bottom pane shows an edited aeration table)

Add an Aeration table

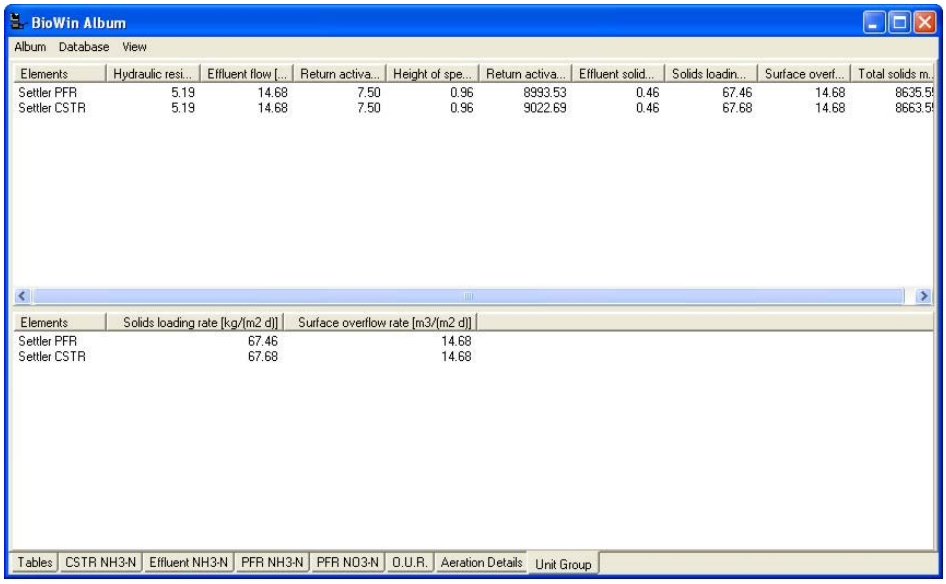
1. Right-click on the album pane where you wish to add the aeration table.
2. From the resulting popup menu, select **Aeration details**. The aeration table will be added to the album pane with the default parameters and variables shown.

Unit Group Table

One problem with ordinary tables is that they may only contain information for parameters that are common to all types of elements. For example, with an ordinary table you cannot select to display **Solids Loading Rate** for a clarifier element. This is because **Solids Loading Rate** is a parameter that is unique to clarifier elements.

Unit group tables give you the flexibility to generate tables for elements that have unique parameters, such as clarifiers. Once you have added a unit group table, you may add additional compounds to it or remove existing ones if you wish.

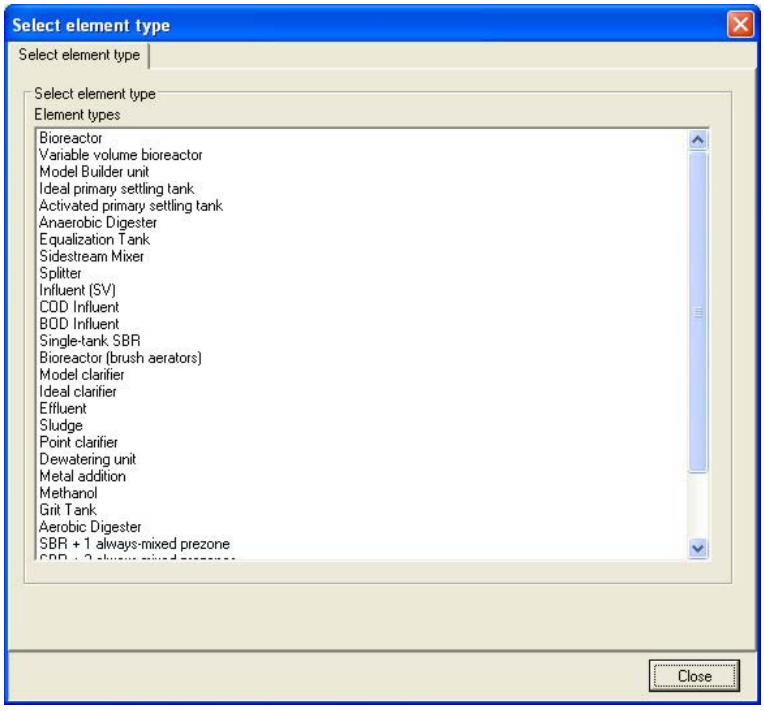
A two-pane album page with example unit group tables is shown below.



Example unit group tables (top pane shows a default unit group table for clarifier element, bottom pane shows an edited unit group table for clarifier element)

Add a Unit Group Table

1. Right-click on the album pane where you wish to add the unit group table.
2. From the resulting popup menu, select **Unit group table**. You will then be presented with the **Select element type** dialog shown below:



Dialog used to select the type of element to include in the unit group table

3. In the **Element type** list, click on the type of element that you want to include in your unit group table.
4. Click the **Close** button to generate the unit group table. You may now edit the table to add additional information if you wish.

Album Chart Sub-Menu

This section provides information about the menu commands available in the **Chart** sub-menu of the pop-up menu that results from right-clicking on a chart in the BioWin album. Note that some of these commands also may be accessed by right-clicking on a table or element information display, but in this case, they will be accessible directly from the resulting pop-up menu as opposed to a sub-menu.

Copy

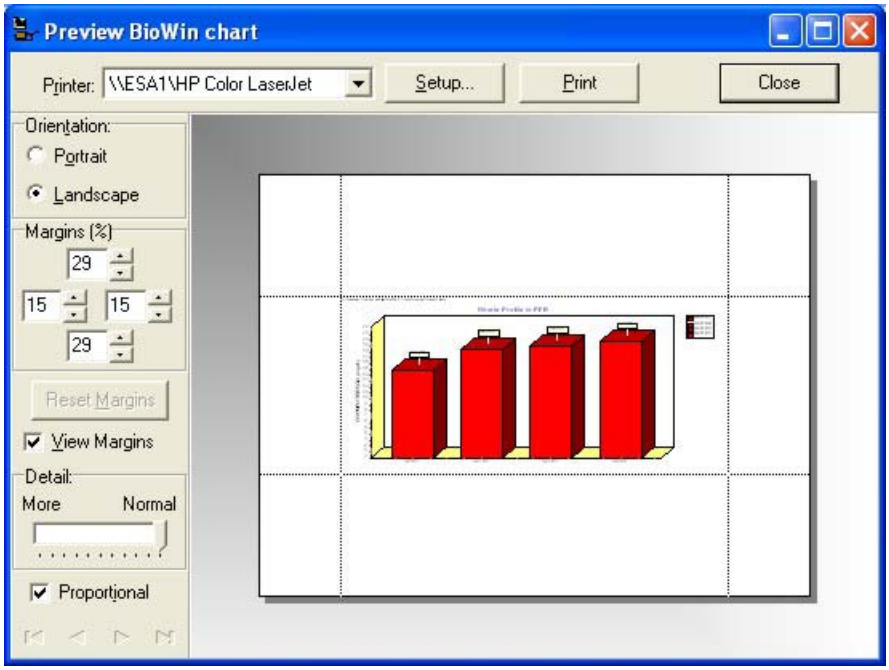
Use this command to place a copy of the current display (i.e. chart, table, element summary, aeration table) on the Windows clipboard. This will make the display available for pasting into another application such as a word processor.

Add To Notes

When this command is invoked for an element information display, table, or aeration table display, BioWin will place a tab-delimited text version of the table into the Simulation Notes editor. If this command is invoked for a chart display, you will be prompted to save the chart file in a graphics file format of your choice, and the file location path will be placed into the Simulation Notes editor for your reference.

Print…

This command will invoke the album display print dialog box, shown below:



Dialog box used for printing album charts, tables, or element information displays

Use the **Printer** drop list box to select the printer you want to use for printing. The **Printer Setup…** button will open the printer setup dialog box which will allow you to access the printer's properties, set paper size, page orientation, and a number of other printer options. The **Print** button will send the print job to the printer and the printout will match the preview which is shown. The **Close** button closes this dialog box and returns you to the album.

Using the **Paper Orientation** group, specify whether you want the printing to be done on a **Portrait** or **Landscape** page. The print preview gives you an idea of what the printout will look like under each format.

If you do not wish to see the size of the margins for your print job, you may de-select the box labeled **View Margins**. You can control the margins using three different methods:

1. Using the **Margins (%)** Spin Edits, you can adjust each margin as you like. The four Spin Edit boxes each control the margin that shares its position, that is, the top Spin Edit controls the top margin, the bottom Spin Edit controls the bottom margin, and so on. When you change a value, you will see changes in the print preview accordingly.
2. You may drag each margin using the mouse. Position the mouse cursor over the margin you wish to adjust until the horizontal or vertical resize cursor appears. Click the mouse button, hold it, and drag the margin to the position you wish it to occupy. Notice that when you finish dragging it, the values in the **Margins (%)** Spin Edits will have been updated.

Resize Cursors

Drag Print Object

3. By moving the object to be printed around on the page. When the mouse cursor takes the form of a hand, you may click and drag the entire object around on the page until it is in the desired position. Notice that when you finish dragging it, the values in the **Margins (%)** Spin Edits will have been updated.

If after applying any one of these methods of adjusting margins you wish to reset the margins to the default values, you may do so by clicking the **Reset Margins** button.

You can use the **Chart Detail** scroll bar to adjust the text size on your print job. Sliding the scroll towards **More** decreases the size of text on your chart and gives greater prevalence to the chart on the printout. Sliding the scroll towards **Normal** increases the size of the text on your chart and gives less prevalence to the chart on the printout.

Note: If you are having trouble with a table print preview not fitting into the margins, ensure that you have a True Type font (font styles with a TT after their name) selected in your **Project | Current Project Options - Drawing board options tab**.

Export…

This command allows you to save the current chart in a number of different graphic formats. When you choose **Export…** from the **Display** sub-menu, you will be presented with a dialog box that has the same functionality as the File Save dialog box.

In the **Save as type:** drop list box, you may choose from one of the many different graphic formats supported by BioWin. When you have specified the file name, location, and type, click the **Save** button. For some formats (e.g. JPEG), this will open a dialog that allows specification of further options relevant to the selected graphic format. Note that if you want to be able to import the chart into another BioWin project, you should export it in **Tee Chart** (*.TEE) format.

Import…

This command allows you to import a chart that you have previously created and exported using BioWin. This command is different from the **Album|Open pages…** command in that it may be used to add individual charts to any album page, while the **Album|Open pages…** command is used to open pages containing charts, tables, and element summaries. When you choose **Import…** from the **Display** sub-menu, you will be presented with a dialog box that has the same functionality as the File Open dialog box.

Note that only charts that have been exported in **Tee Chart** (*.TEE) format are available for importing. Once you have located the chart you wish to import, click the **Open** button to import it to the current chart display. The imported chart will replace the current chart display. Any series that exist on the imported chart will not be linked to the current database – they essentially are inactive drawings on the chart. However, the **Import…** command is useful if you wish to compare values from another project or set of simulations to current values, if you wish to use similar chart and series formatting, or if you wish to import charts from other applications which use the **Tee Chart** (*.TEE) format.

Change to Table…

This command will change the current display from a chart or an element summary to a table (or a new table if the current display is already a table). Values from the current chart, element summary, or table will not be placed in the table; this command is simply for changing the type of display in the current pane.

Change to Element info…

This command will change the current display from a chart or table to an element summary (or a different type of element summary if the current display is already an element summary). Values from the current chart, element summary, or table will not be placed in the element summary; this command is simply for changing the type of display in the current pane.

Change to Chart

This command will change the current display from a table or element summary to a chart. Values from the current element summary or table will not be placed in the chart; this command is simply for changing the type of display in the current pane.

Delete current…

This command will delete the current display from an album pane. This command is useful if you wish to delete one information display from a multi-pane album page without deleting or changing the displays on the other panes of the page.

Creating Charts & Adding Series

Charts in BioWin

Charts are used in BioWin to display various types of series. It is possible to change the formatting of certain chart properties. For more information see "*Chart Formatting Procedures*".

Add a Chart To The Album

1. Open the album by selecting **View|Album** in the BioWin main window.
2. Add a new page to the album by selecting **Album|Add page…**
3. From the **Select new album page** gallery, select the desired layout.
4. Right-click on the pane that you want the chart to occupy.
5. Choose **Chart** from the resulting pop-up menu.

Series Styles in BioWin

A series may be any one of a variety of styles including:

- Line
- Fast Line
- Point
- Bar (Horizontal and Vertical)
- Area
- Pie
- Surface

Users will find that certain series styles are best suited to specific series types. For example, applying the Pie series style to time series data is probably not as effective as using the Line or Fast Line series style. Each series style has its own set of unique formatting options. For further information on changing these formatting options, see "*Series Formatting Procedures*".

Series Available From The Album

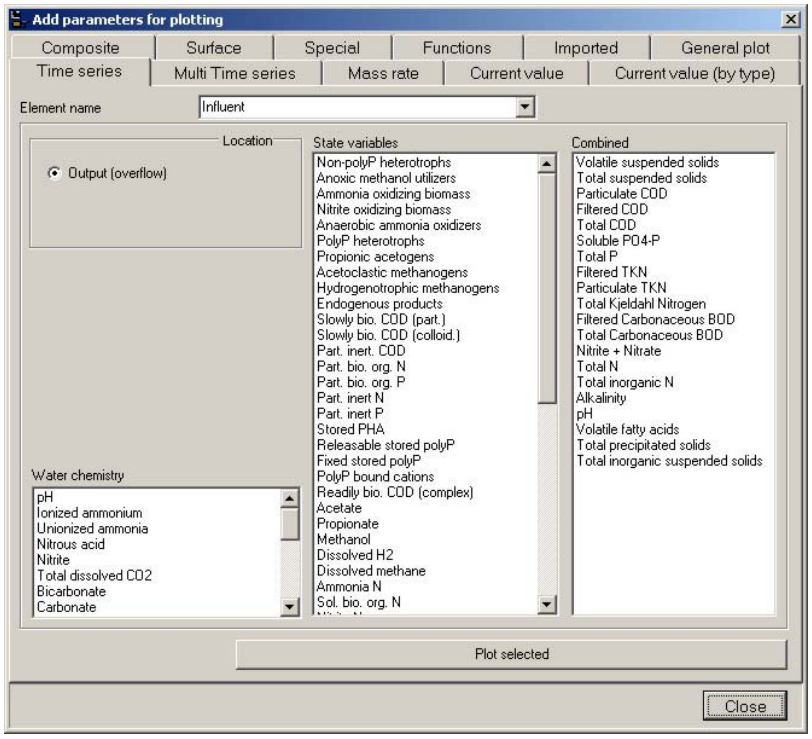
BioWin offers users a number of different series types that may be used to plot data. This section outlines information and procedures related to series types that may be added to a chart from the album.

Time Series (Album)

This series type is used for time series analysis of data. It is unique in that the user may specify the element location (i.e. Input, Output, Underflow) from which data for the series are obtained. Also, this is the only type of series for which you may plot element-specific information (e.g. Solids Loading Rate for settling tank elements, Oxygen Utilization Rates for bioreactor elements, etc).

Add a Time Series From the Album

1. Right-click on an album chart and click **Add Series** in the resulting pop-up menu.
2. On the **Time series** tab, select the element you wish to plot a variable for from the **Element name** drop list box.



The time series (from album) dialog box

3. Select the location in the element where you wish to obtain the plotting data from (i.e. Input, Output {overflow}, Underflow) using the **Location** radio button group.
4. Choose a parameter to plot from the **Element specific**, **Water Chemistry**, **State variables**, or **Combined** list boxes. If you want to add more than one parameter from a given group, you may do so. To select a contiguous group, click the first parameter of the group, and

while holding the **Shift** key, click the last parameter of the group. To select non-contiguous parameters, hold the **Ctrl** key and click the desired parameters in succession.

5. Click the **Plot selected** button.
6. From the **Time series gallery**, choose the desired series style that you wish to apply. If you want your chart to have a three-dimensional appearance, ensure that the box labeled **3D** is checked and click **OK**.
7. Click the **Close** button to finish.

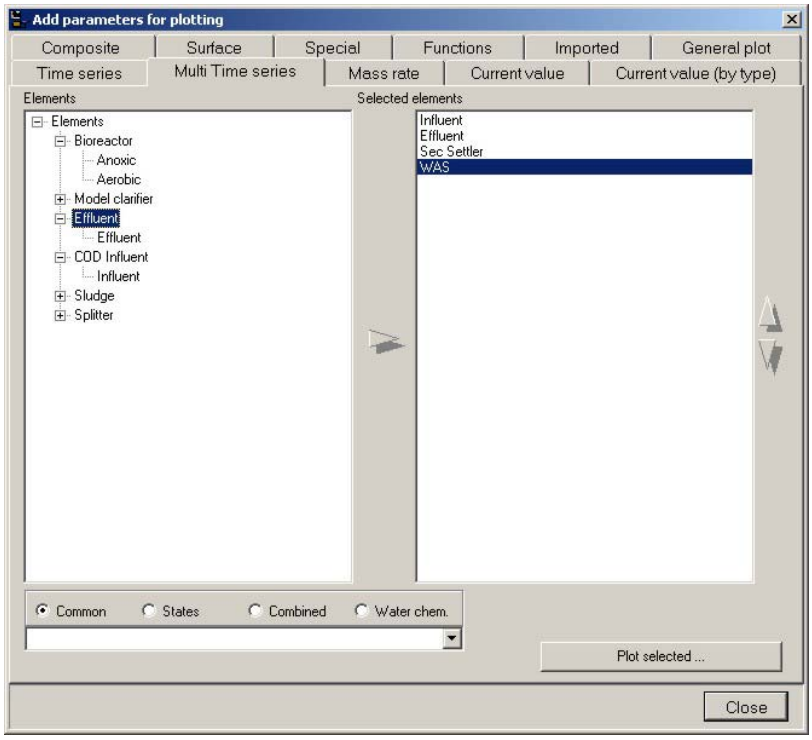
Note: You may only select multiple parameters in one list box at a time. If you want to plot more than one parameter and they are not in the same list box, you must repeat steps 3-6 for each parameter. Some features of the dialog box described above will only appear for certain elements. For example, the underflow location option only will appear for elements that have underflows (such as settling tank elements). Also, the element specific list box only will appear for elements that have unique data types (such as bioreactors).

Multi Time Series

Multi time series provide a quick means with which to plot a chosen compound in a number of different elements. For example, you quickly can generate a group of series that show the ammonia concentration in each bioreactor. Note that for this series type the data for plotting are taken from the main output of the element (for example, the overflow of a settler).

Add a Multi Time Series

1. Right-click on an album chart and click **Add Series** in the resulting pop-up menu.
2. Click on the **Multi Time series** tab of the **Add Series** dialog box.
3. In the **Elements** tree view list, select the elements that you wish to include in the profile plot.



The multi time series dialog

- If you wish to include all the elements from a group (for example, all the Bioreactors), click on the group heading and then click the right-pointing arrow to move them all to the **Selected elements** list.
- If you wish to include only certain elements from a group (or groups), then click on the plus sign (+) next to the group heading to expand it, click on the specific element you want to include, and then click the right-pointing arrow to move it to the **Selected elements** list.

4. If you want to change the order in which the **Selected elements** will appear in the multi time series plot, move the elements around by clicking on them and clicking the up/down arrows.
5. Select the compound that you wish to plot from the **Compound** drop list box.
6. Click the **Plot selected** button.
7. From the **Time series gallery**, choose the desired series style that you wish to apply to your profile. If you want your chart to have a three-dimensional appearance, ensure that the box labeled **3D** is checked and click **OK**.
8. Click the **Close** button to finish.

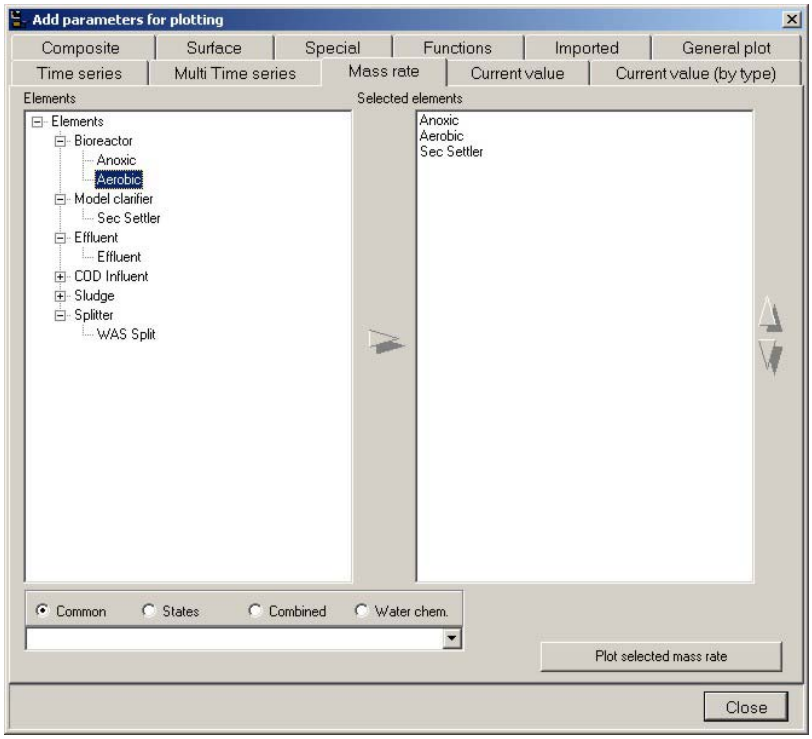
Note: You may repeat steps 5-7 if you wish to generate multi time series for more than one compound on a chart.

Mass Rate Series

Mass rate series allow you to plot the mass rate against time for state variables and compounds. Note that for this series type the data for plotting are taken from the main output of the element (for example, the overflow of a settler).

Add a Mass Rate Series

1. Right-click on an album chart and click **Add Series** in the resulting pop-up menu.
2. Click on the **Mass Rate** tab of the **Add Series** dialog box.
3. In the **Elements** tree view list, select the elements that you wish to include in the mass rate plot.



The mass rate series dialog

- If you wish to include all the elements from a group (for example, all the Bioreactors), click on the group heading and then click the right-pointing arrow to move them all to the **Selected elements** list.
- If you wish to include only certain elements from a group (or groups), then click on the plus sign (+) next to the group heading to expand it, click on the specific element you want to include, and then click the right-pointing arrow to move it to the **Selected elements** list.

4. If you want to change the order in which the **Selected elements** will appear in the mass rate plot, move the elements around by clicking on them and clicking the up/down arrows.

5. Select the compound that you wish to plot from the drop list box below the element list.
6. Click the **Plot selected mass rate** button.
7. From the **Time series gallery**, choose the desired series style that you wish to apply to your profile. If you want your chart to have a three-dimensional appearance, ensure that the box labeled **3D** is checked and click **OK**.
8. Click the **Close** button to finish.

Note: You may repeat steps 5-7 if you wish to generate mass rate series for more than one compound on a chart.

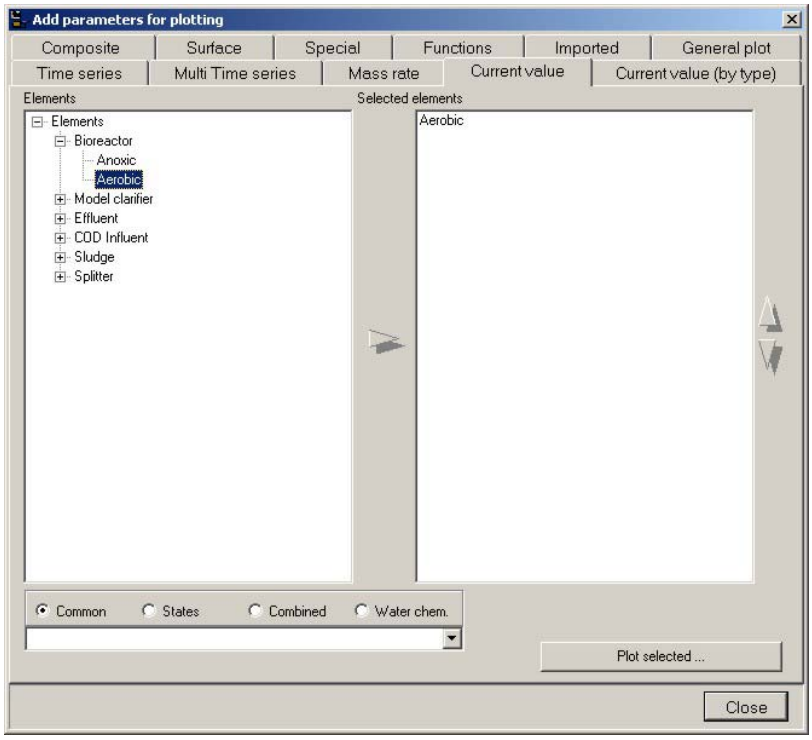
Current Value

For a group of selected elements (or a single element), a current value series will show the most recent (with respect to simulation start time) value of a chosen compound. During a dynamic simulation, the series will be updated and redrawn after each data interval. For a steady-state simulation, the series will represent the final steady-state values.

Current value plots are best suited to a specific group of the available series styles, such as Bar, Pie, and Area. As such, the choice of styles in the gallery is limited to these.

Add a Current Value Series

1. Right-click on an album chart and click **Add Series** in the resulting pop-up menu.
2. Click on the **Current value** tab of the **Add Series** dialog box.
3. In the **Elements** tree view list, select the element(s) that you wish to include in the current value plot.



The current value series dialog

- If you wish to include all the elements from a group (for example, all the bioreactors), click on the group heading and then click the right-pointing arrow to move them all to the **Selected elements** list.
- If you wish to include only certain elements from a group (or groups), then click on the plus sign (+) next to the group heading to expand it, click on the specific element you want to include, and then click the right-pointing arrow to move it to the **Selected elements** list.

4. If you want to change the order in which the **Selected elements** will appear in the current value plot, move the elements around by clicking on them and clicking the up/down arrows.
5. Select the compound that you wish to plot from the **Compound** drop list box.
6. Click the **Plot selected** button.
7. From the **Current value series gallery**, choose the desired series style that you wish to apply. If you want your chart to have a three-dimensional appearance, ensure that the box labeled **3D** is checked and click **OK**.
8. Click the **Close** button to finish.

Note: If you wish you may have more than one current value series on a chart. Simply repeat steps 5-7 for each compound that you wish to have a current value series for. Keep in mind however that this works best for Bar and Area charts, since Pie charts are drawn over top of one another.

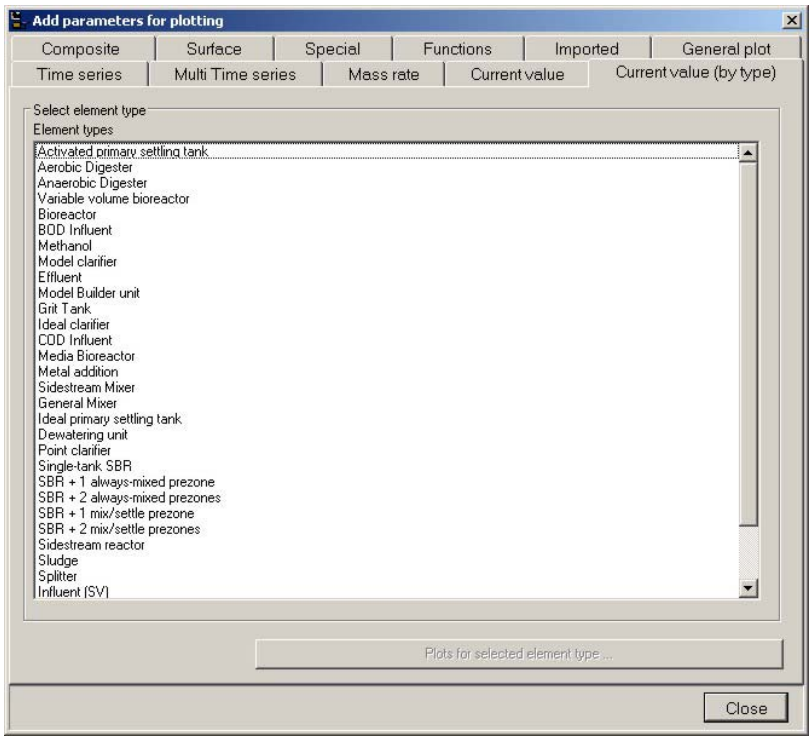
Current Value (by type)

For a group of selected elements (or a single element) from a chosen element class (e.g. bioreactor, clarifier), a current value (by type) series will show the most recent (with respect to simulation start time) value of a chosen compound. During a dynamic simulation, the series will be updated and redrawn after each data interval. For a steady-state simulation, the series will represent the final steady-state values.

The compounds that may be plotted in an ordinary current value series are limited to those that are common to all element types. For example, in an ordinary current value series, you may not plot OUR for a bioreactor, as this parameter is specific to bioreactor-class elements. The current value (by type) series overcomes this limitation by allowing you to first choose the type of element that the plot data are from, and then offering a customized list of compounds based on the element type. For example, if you select a clarifier element, you will be able to plot current values of **Solids Loading Rate**.

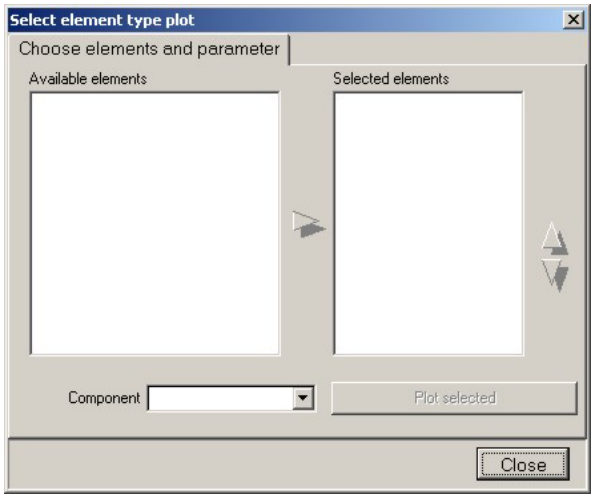
Add a Current Value (by type) Series

1. Right-click on an album chart and click **Add Series** in the resulting pop-up menu.
2. Click on the **Current value (by type)** tab of the **Add Series** dialog box.
3. In the **Element types** list, select the element type that you wish to include in the current value (by type) plot.



The current value(by type) series dialog

4. Click the **Plots for selected element type…** button to open a dialog that allows you to choose the elements and compounds for your plot. This dialog is shown below:



Dialog used for selecting elements and element-specific compounds

5. If you want to change the order in which the **Selected elements** will appear in the current value plot, move the elements around by clicking on them and clicking the up/down arrows.
6. Select the compound that you wish to plot from the **Component** drop list box.
7. Click the **Plot selected** button to plot a **Bar** series. You may change the series style after finishing your plot if you wish.
8. Click the **Close** button to finish.

Note: If you wish you may have more than one current value (by type) series on a chart. Simply repeat steps 6-8 for each compound that you wish to have a current value series for.

Composite Sample Series

A composite sample series can be used to plot composite sampling data for a given parameter in one or more elements. You may specify the sampling period (e.g. a daily composite, a weekly composite, etc.), any offsets that will take place in the sampling period, and whether you want the data to be flow weighted average or a simple average.

An important concept is the **Sampling period**. The sample is taken and plotted at the **end** of the sampling period. For example, if you have a 1 day period (i.e. a daily composite), then the first sample will be taken and plotted at the end of the first 24 hour period, and at the end of each 24 hour period thereafter. Note that the sampling period must be greater than the database data interval, otherwise the sample will not be a composite.

Another important concept is the **Offset**. This is the amount by which you step into the sampling period when you start simulating. This concept is perhaps best illustrated by an example. Say you have a 24 hour period, with a 16 hour offset, and your simulation start time is 12:00 AM. Since you are stepping into the sampling period at the 16 hour point, this means that the first point will be sampled and plotted in 8 hours (recall from the discussion of sampling period above that samples are always taken and plotted at the end of the sampling period). This means that your

first sample will be taken and plotted at 8:00 AM on the first day, and then at 8:00 AM each day thereafter for the duration of the simulation.

Note: It should be noted that composite sampling data are not added to BioWin's database.

You may choose to plot flow weighted averages of your samples, or simple averages. If you choose to have flow weighted average values plotted, the following formula will be used for **n** points (where **Q** is flow and **C** is concentration):

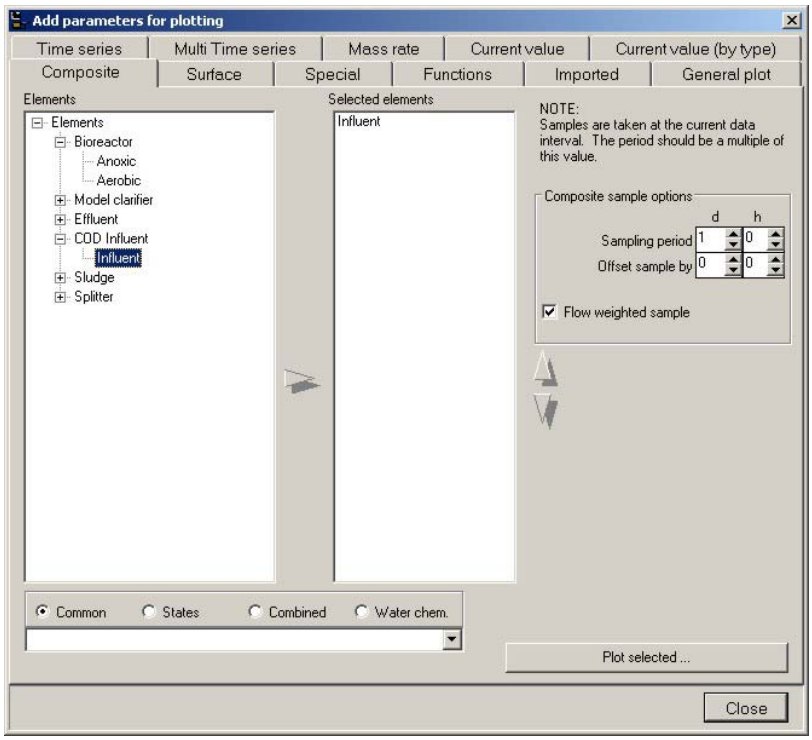
$$F.W.Avg. = \frac{\sum_{i=1}^{n} Q_i C_i}{\sum_{i=1}^{n} Q_i}$$

If you choose to have simple average values plotted, then the following formula will be used:

$$Avg. = \frac{\sum_{i=1}^{n} C_i}{n}$$

Add a Composite Series

1. Right-click on an album chart and click **Add Series** in the resulting pop-up menu.
2. Click on the **Composite** tab of the **Add Series** dialog box.
3. In the **Elements** tree view list, select the element(s) for which you wish to plot composite data.



The composite series dialog.

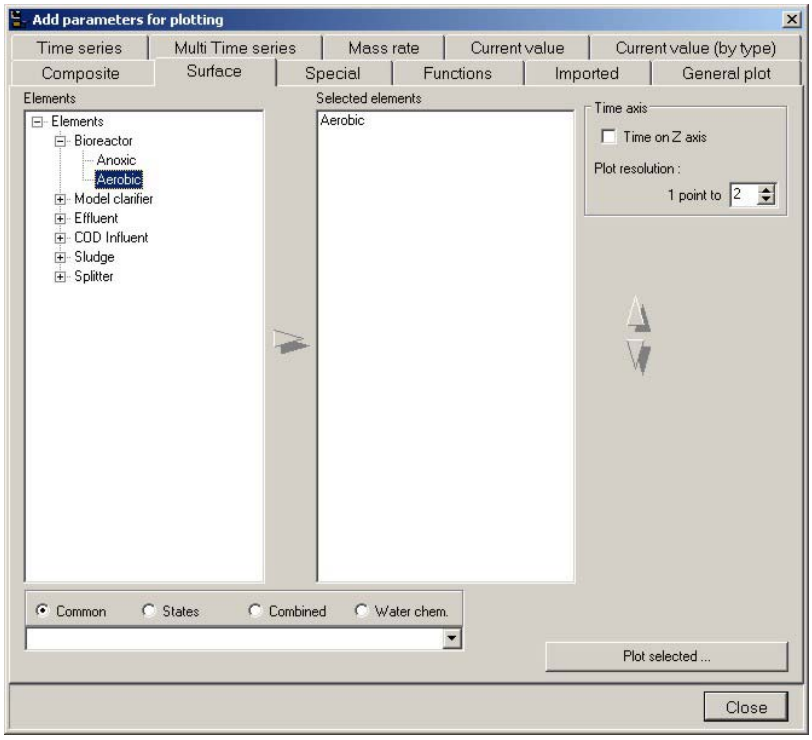
4. You may use the up/down arrows next to the **Selected elements** list to change the order in which the series are plotted if you wish.
5. From the **Compound** drop list, select the parameter for which you wish to plot composite samples.
6. Select your **Sampling period** (recall that data are sampled and plotted at the *end* of the sampling period).
7. You may select an offset at the beginning of the simulation using the **Offset sample by** control (recall that this is the amount that you step into the sampling period – see above discussion).
8. Click the **Plot selected** button. From the series gallery, select the series style you want to use. For composite plots, **Point Series** are useful.
9. Click **OK** to close the series gallery.
10. Click **Close** to finish.

Surface Series

Surface series can be used for spatial profiles of a given compound. For example, say you have a configuration with three bioreactors, and you want to show the ammonia profile over the bioreactors. The ammonia concentration would be plotted on the Y (vertical) axis, the bioreactor number would be plotted on the X (horizontal) axis, and time would be plotted on the Z axis.

Add a Surface Series

1. Right-click on an album chart and click **Add Series** in the resulting pop-up menu.
2. Click on the **Surface** tab of the **Add Series** dialog box.
3. In the **Elements** tree view list, select the element(s) that you wish to include in the surface plot.



The surface series dialog

- If you wish to include all the elements from a group (for example, all the bioreactors), click on the group heading and then click the right-pointing arrow to move them all to the **Selected elements** list.
- If you wish to include only certain elements from a group (or groups), then click on the plus sign (+) next to the group heading to expand it, click on the specific element you want to include, and then click the right-pointing arrow to move it to the **Selected elements** list.

4. If you want to change the order in which the **Selected elements** will appear in the surface plot, move the elements around by clicking on them and clicking the up/down arrows.
5. Select the horizontal axis that you want time to be recorded on. If you check the box labeled **Time on Z axis**, then time will be recorded on the Z axis (the axis that goes into the screen). If this box is left unselected, then time will be recorded on the X axis (the horizontal axis).
6. Set your **Plot resolution**. If you want a fine grid, decrease the value in the spin edit box. If you want a coarse grid, increase the value. Note that fine grids look better, but will increase demand on system resources.
7. Select the compound that you wish to plot from the **Compound** drop list box.
8. Click the **Plot selected** button.
9. Click the **Close** button to finish.

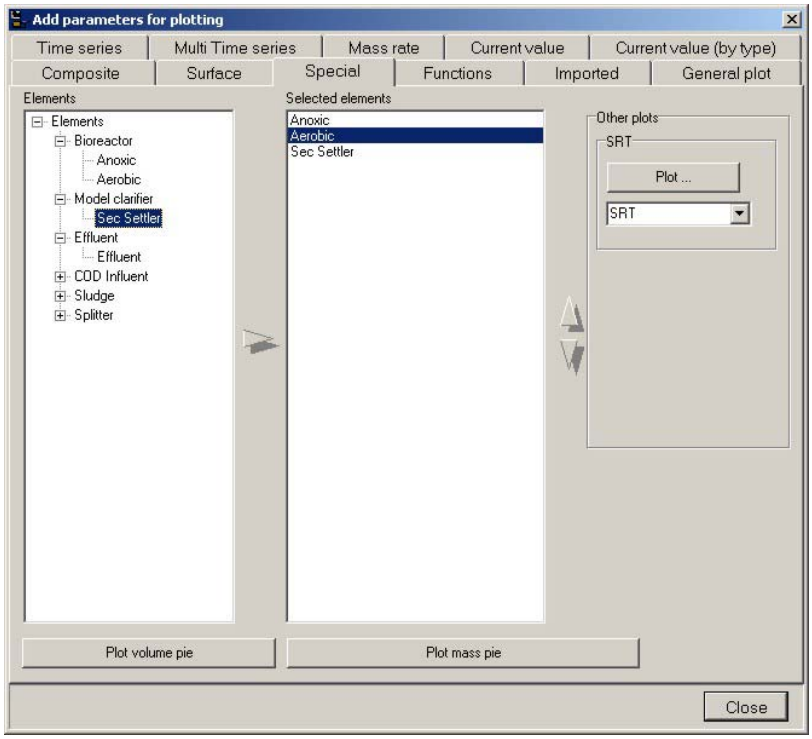
Special Series (Album)

BioWin comes with some plots that are generated semi-automatically. These include process volume, mass fraction, and SRT plots. For process volume and mass fraction plots, you only are required to select the elements that you want included in the plots. BioWin then performs the necessary calculations and generates the plot. For SRT plots, you simply select the calculated SRT. that you want to plot dynamically. You then may customize the appearance of the chart using BioWin's powerful chart and series formatting tools.

Add a Special Series From the Album

If you want to add a process volume or mass fraction series:

1. Right-click on an album chart and click **Add Series** in the resulting pop-up menu.
2. Click on the **Special** tab of the **Add Series** dialog box.
3. In the **Elements** tree view, select the element(s) that you wish to include in the special plot.



The special series (from album) dialog

- If you wish to include all the elements from a group (for example, all the bioreactors), click on the group heading and then click the right-pointing arrow to move them all to the **Selected elements** list.

- If you wish to include only certain elements from a group (or groups), then click on the plus sign (+) next to the group heading to expand it, click on the specific element you want to include, and then click the right-pointing arrow to move it to the **Selected elements** list.

4. If you want to change the order in which the **Selected elements** will appear in the special plot, move the elements around by clicking on them and clicking the up/down arrows.
5. When you are satisfied with the order of the included elements, click on the **Plot volume pie** button to generate a pie series showing the volume fraction of each selected element, or click on the **Plot mass pie** button to generate a pie series showing the mass fraction of each selected element.
6. Click the **Close** button to finish.

If you want to add a time series plot of a calculated SRT:

1. Right-click on an album chart and click **Add Series** in the resulting pop-up menu.
2. Click on the **Special** tab of the **Add Series** dialog box.
3. In the **SRT** group of the **Other plots** section, select the calculated SRT that you want to plot, and click the **Plot…** button.
4. From the **Time series gallery**, choose the desired series style that you wish to apply. If you want your chart to have a three-dimensional appearance, ensure that the box labeled **3D** is checked and click **OK**.
5. Click the **Close** button to finish.

Imported Series

BioWin offers the functionality of plotting imported data. This is useful particularly for comparing simulation results to observed data (if you are calibrating BioWin, for example). There are two types of series you can plot using imported data:

1. Final value
2. Time series (X-Y)

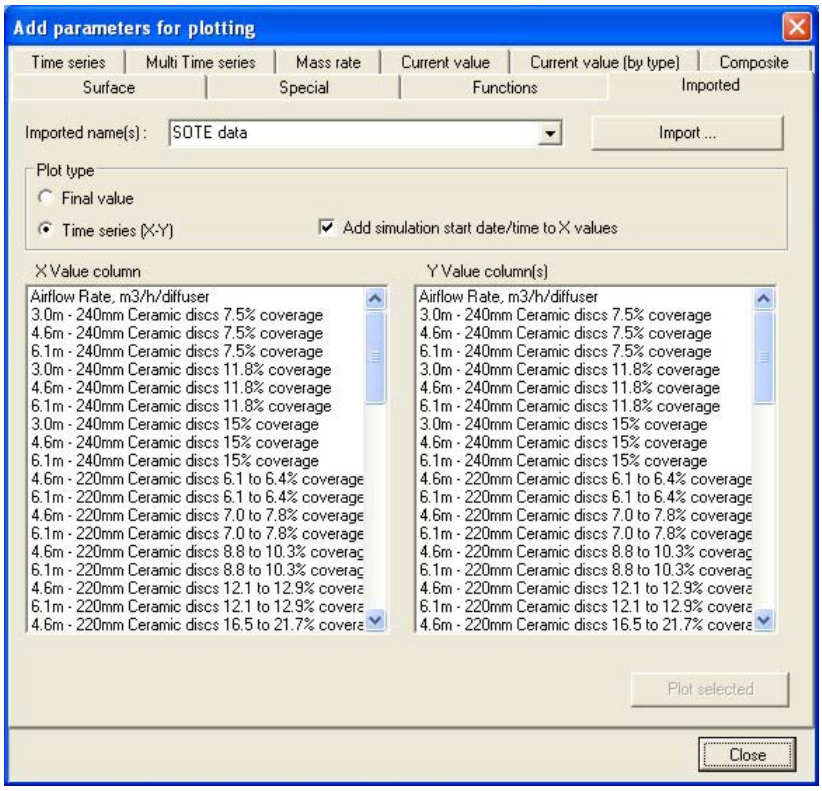
A **Final value** series allows you to plot the last value from the column(s) of data that you select. Therefore, this series type is similar to a current value series for imported data and is suited to styles such as bar and pie series.

A **Time series** allows you to plot all of the values in an imported data column against all of the values in another imported data column (usually the imported time column). Therefore, this series type is best suited to styles such as line and point series.

Add an Imported Series

1. Right-click on an album chart and click **Add Series** in the resulting pop-up menu.
2. Click on the **Imported** tab of the **Add Series** dialog box.

3. Using the **Imported Name(s)** drop list box, select the name of a file or block of data (from the clipboard) you have imported to the database.



The imported series dialog

4. If you want to import a file to the database from this dialog box, you may do so by clicking the **Import…** button. For more information on importing data from a file, please see **Importing Data**.
5. Select the **Plot type**.

If you select a **Final value** series:

1. From the **Y Value column(s)** list, select the columns that you want to plot the final value of. To select a contiguous group of columns, click the first column of the group, and while holding the **Shift** key, click the last column of the group. To select non-contiguous columns, hold the **Ctrl** key and click the desired columns in succession.
2. Click the **Plot selected** button.
3. From the **General series gallery**, choose the desired series style that you wish to apply. If you want your chart to have a three-dimensional appearance, ensure that the box labeled **3D** is checked and click **OK**.
4. Click the **Close** button to finish.

If you select a **Time series**:

1. From the **X Value column** list, select the column that you want to use as the independent variable in the plot. If you have selected a column containing time values, then checking the box labeled **Add simulation start date/time to X values** will synchronize the imported time values to the original database time values. This option will allow you to plot a time series generated using imported data on the same axes as a time series that uses non-imported data.
2. From the **Y Value column(s)** list, select the columns that you want to use as independent variables. To select a contiguous group of columns, click the first column of the group, and while holding the **Shift** key, click the last column of the group. To select non-contiguous columns, hold the **Ctrl** key and click the desired columns in succession. Each column that you select will yield a series on the chart.
3. Click the **Plot selected** button.
4. From the **General series gallery**, choose the desired series style that you wish to apply. If you want your chart to have a three-dimensional appearance, ensure that the box labeled **3D** is checked and click **OK**.
5. Click the **Close** button to finish.

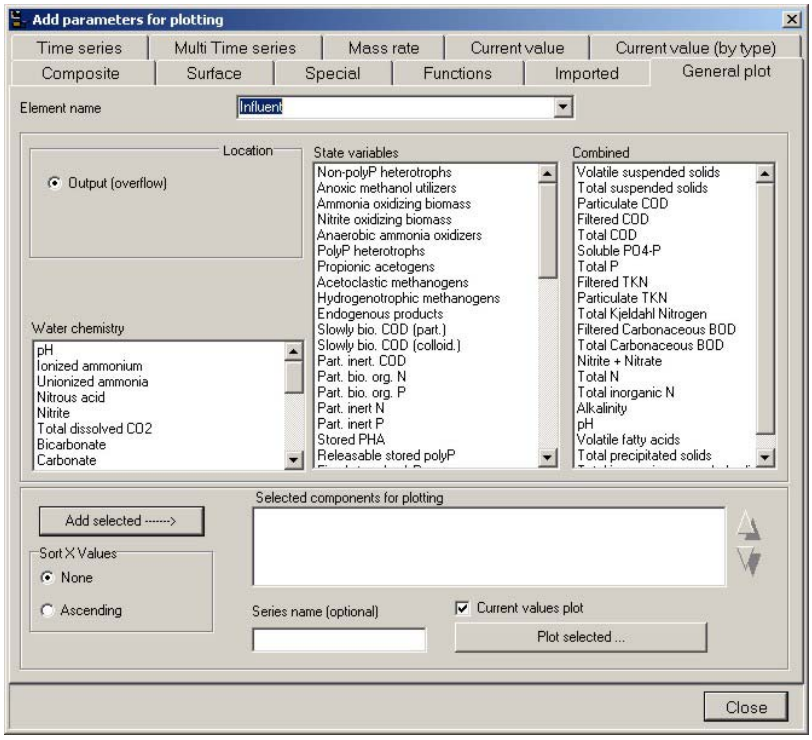
General Plot (Album)

This tab can be used to plot current value series or X-Y "scatter" plots. It is unique in that the user may specify the element location (i.e. **Input**, **Output**, **Underflow**) from which data for the series are obtained. From this tab you may also plot element-specific information (e.g. **Solids Loading Rate** for settling tank elements, **Oxygen Utilization Rates** for bioreactor elements, etc) in either a "**current value**" or "**X-Y scatter**" type plot.

Note: Parameters selected for plotting will automatically be added to the list of "Monitored" items.

Add a General "Current Value" Series From the Album

1. Right-click on an album chart and click **Add Series** in the resulting pop-up menu.
2. On the **General Plot** tab, select the element you wish to plot a variable for from the **Element name** drop list box.



The general plot (from album) dialog box

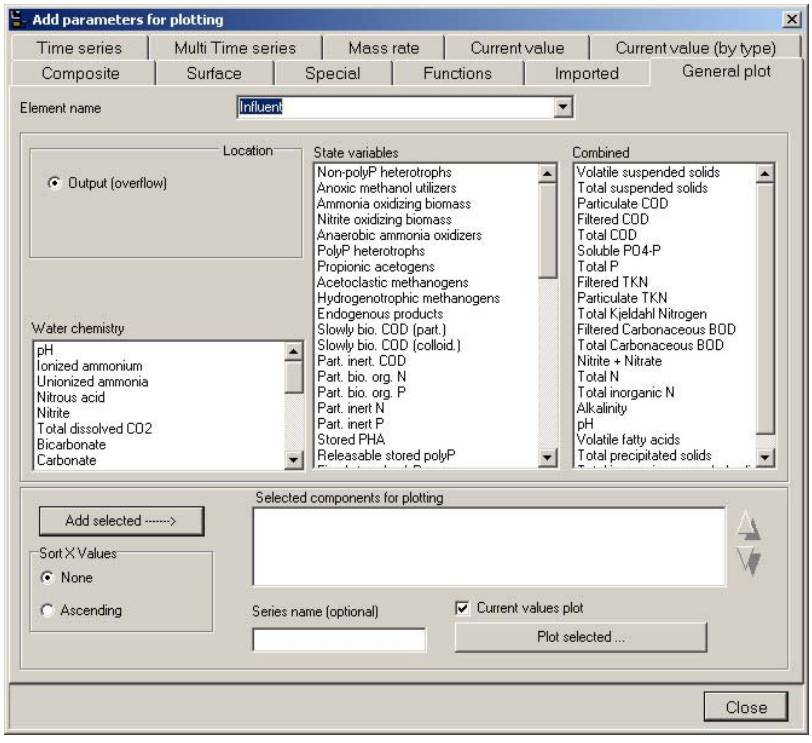
3. Select the location in the element where you wish to obtain the plotting data from (i.e. **Input, Output** {**overflow**}, **Underflow**) using the Location radio button group.
4. Choose a parameter to plot from the **Element specific**, **Water Chemistry**, **State variables**, or **Combined** list boxes. If you want to add more than one parameter from a given group, you may do so. To select a contiguous group, click the first parameter of the group, and while holding the **Shift** key, click the last parameter of the group. To select non-contiguous parameters, hold the **Ctrl** key and click the desired parameters in succession.
5. Click the **Add** selected à button.
6. The item(s) that you have selected are listed in the **Selected components for plotting** list box. You can change the order of the parameters using the "up" and "down" arrows on the right of the list box, and delete a parameter by selecting it and then hitting the **Del** key.
7. To add additional parameters for the selected element repeat steps 3 through 6. To add parameters for a different element repeat steps 2 through 6.
8. Type a name for the series in the edit box directly below the **Selected components for plotting** list box.
9. Make sure that the **Current values** plot box is checked.
10. Press the **Plot selected …** button.
11. From the **General series** gallery, choose the desired series style that you wish to apply. If you want your chart to have a three-dimensional appearance, ensure that the box labeled **3D** is checked and click OK.

12. To add another **Current value** series repeat steps 2 through 11.
13. Click the **Close** button to finish.

Note: You may only select multiple parameters in one list box at a time. If you want to plot more than one parameter and they are not in the same list box, you must repeat steps 3-6 for each parameter. Some features of the dialog box described above will only appear for certain elements. For example, the underflow location option only will appear for elements that have underflows (such as settling tank elements). Also, the element specific list box only will appear for elements that have unique data types (such as bioreactors).

Add a General "X-Y Scatter" Series From the Album

1. Right-click on an album chart and click **Add Series** in the resulting pop-up menu.
2. On the **General Plot** tab make sure that the **Current values** plot box is not checked.
3. In an X-Y scatter plot the first parameter selected is plotted on the X axis and the second parameter is plotted on the y axis. Any other parameters listed in the **Selected components for plotting** list box are not used. You can change the order of the parameters using the "**up**" and "**down**" arrows on the right of the list box, and delete a parameter by selecting it and then hitting the **Del** key.
4. Select the element you wish to plot a variable for from the **Element name** drop list box.



The general plot (from album) dialog box

5. Select the location in the element where you wish to obtain the plotting data from (i.e. Input, Output {overflow}, Underflow) using the **Location** radio button group.
6. Choose a parameter to plot from the **Element specific, Water Chemistry, State variables,** or **Combined** list boxes.
7. Click the **Add selected →** button.
8. To add a second parameter for the selected element repeat steps 5 through 7. To add parameters for a different element repeat steps 4 through 7.
9. The items that you have selected are listed in the **Selected components for plotting** list box (the axis that will be used for each parameter is shown to the left of the parameter.
10. Type a name for the series in the edit box directly below the **Selected components for plotting** list box.
11. Press the **Plot selected …** button.
12. From the **General series** gallery, choose the desired series style that you wish to apply. If you want your chart to have a three-dimensional appearance, ensure that the box labeled 3D is checked and click OK.
13. To add another X-Y scatter plot series repeat steps 4 through 12.
14. Click the **Close** button to finish.

Note: In an X-Y scatter plot a time history of both of the parameters is required, so when you first add the series there may not be any data to show. Since the parameters involved in the scatter plot are automatically monitored the plot will show when a dynamic simulation is run.

Series Available From The Drawing Board

In the previous section, procedures for adding series to charts from the album view were outlined. This section covers procedures related to charts created directly from the drawing board by right-clicking on elements, and the specialized series that result.

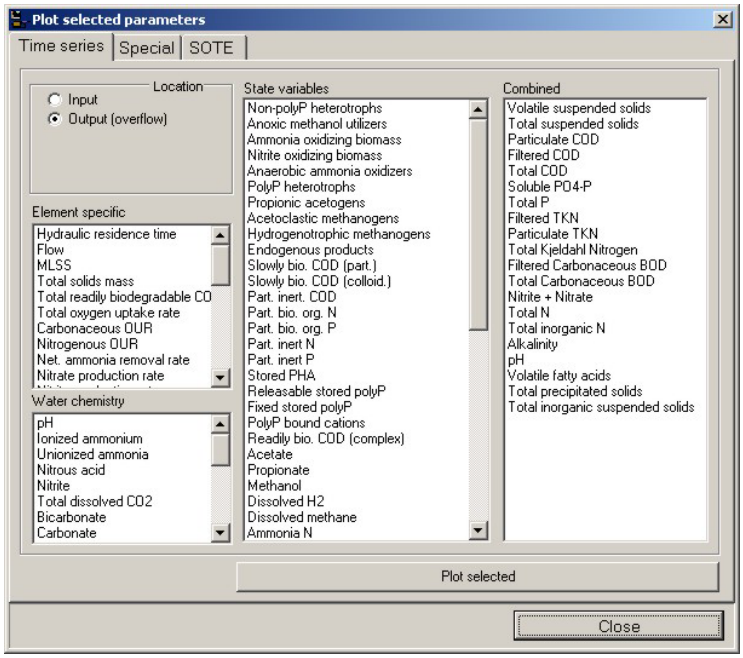
When you add a series from the drawing board, a page with a chart in which to plot the series will be added automatically to the album (even if the album is closed or in the background at the time). If the album was last closed or placed in the background with a page already containing a chart selected, BioWin will ask you if you want to plot the series you are adding from the drawing board in the existing chart. You may agree to this, or by declining choose to have BioWin add a new page with a chart in which to plot the series. Note that if the last selected album page contains multiple panes (where at least one of the panes contains a chart), a chart must have been selected (i.e. clicked on) before the album was closed in order for BioWin to give you the option of adding the drawing board-generated series to that chart. If the chart was not selected upon closing the album, you will not be offered the option of adding the drawing board series to it – BioWin will add a new page with a chart for the drawing board-generated series.

Time Series (Drawing Board)

This series type is used for time series analysis of data. It is unique in that the user may specify the element location (i.e. Input, Output, Underflow) from which data for the series are obtained. Also, this is the only type of series for which you may plot element-specific information (e.g. Solids Loading Rate for settling tank elements, Oxygen Utilization Rates for bioreactor elements, etc.).

Add a Time Series From the Drawing Board

1. In the drawing board, right-click on the element that you wish to create a time series for.
2. From the resulting popup menu, select **Add to album**. Next, select **Chart…** from the resulting flyout. If you are presented with the choice of adding the series to an existing chart, click **Yes** or **No**.
3. On the **Time series** tab, select the location in the element where you wish to obtain the plotting data from (i.e. Input, Output {overflow}, Underflow) using the **Location** radio button group.



The time series (from drawing board) dialog

4. Choose a parameter to plot from the **Element specific**, **Water Chemistry**, **State variables**, or **Combined** list boxes. If you want to add more than one parameter from a given group, you may do so. To select a contiguous group, click the first parameter of the group, and while holding the **Shift** key, click the last parameter of the group. To select non-contiguous parameters, hold the **Ctrl** key and click the desired parameters in succession.
5. Click the **Plot selected** button.
6. From the **Time series gallery**, choose the desired series style that you wish to apply. If you want your chart to have a three-dimensional appearance, ensure that the box labeled **3D** is checked and click **OK**.

7. Click the **Close** button to finish.

Note: You may only select multiple parameters in one list box at a time. If you want to plot more than one parameter and they are not in the same list box, you must repeat steps 3-6 for each parameter. Some features of the dialog box described above will only appear for certain elements. For example, the underflow location option only will appear for elements that have underflows (such as settling tank elements). Also, the element specific list box only will appear for elements that have unique data types (such as bioreactors).

Special Series (Drawing Board)

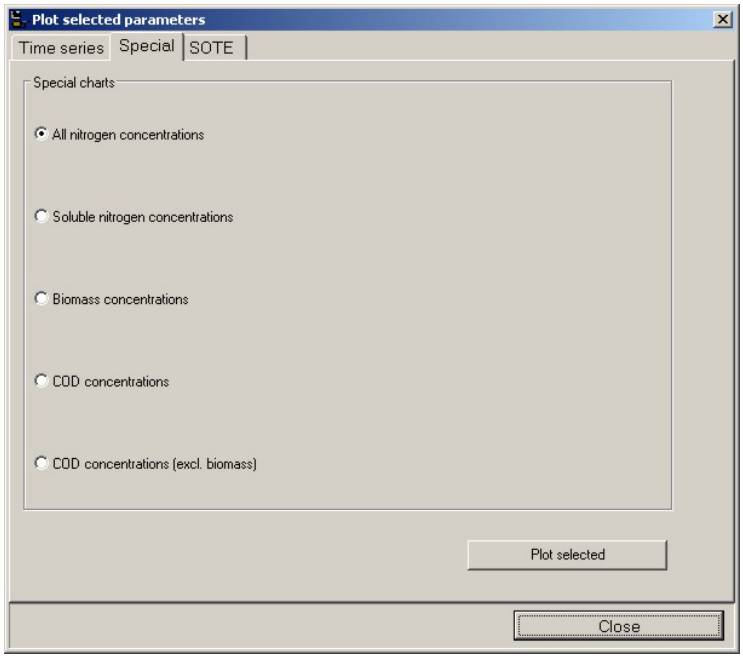
BioWin comes with a collection of special series that are generated automatically. You simply choose the type of special series and BioWin performs the necessary calculations to generate the data required. You may then format the series and chart as you wish. Note that these series are a special form of current value series – that is they display the most recent data with respect to simulation start time.

The available special series are listed here with a description:

Special Series	Description
All nitrogen concentrations	Displays the current concentration of all nitrogen species for the selected element.
Soluble nitrogen concentrations	Displays the current concentration of all soluble nitrogen species for the selected element.
Biomass concentrations	Displays the current concentration of autotrophs, heterotrophs, poly-p heterotrophs, and endogenous residue for the selected element.
COD concentrations	Displays the current COD concentrations (including biomass expressed in terms of COD) for the selected element.
COD concentrations excluding biomass	Displays the current COD concentrations (excluding biomass expressed in terms of COD) for the selected element.

Add a Special Series From the Drawing Board

1. In the drawing board, right-click on the element that you wish to create a special series for.
2. From the resulting popup menu, select **Add to album**. Next, select **Chart…** from the resulting flyout. If you are presented with the choice of adding the series to an existing chart, click **Yes** or **No**.
3. On the **Special** tab, click the radio button next to the special series you want.



The special series (from drawing board) dialog

4. Click the **Plot selected** button.
5. From the **Current value series gallery**, choose the desired series style that you wish to apply. If you want your chart to have a three-dimensional appearance, ensure that the box labeled **3D** is checked and click **OK**.
6. Click the **Close** button to finish.

Note: You may add more than one special series to a chart by repeating steps 3-5. Keep in mind that this will not work well for Pie series since they are drawn over top of one another.

SOTE (%) Series

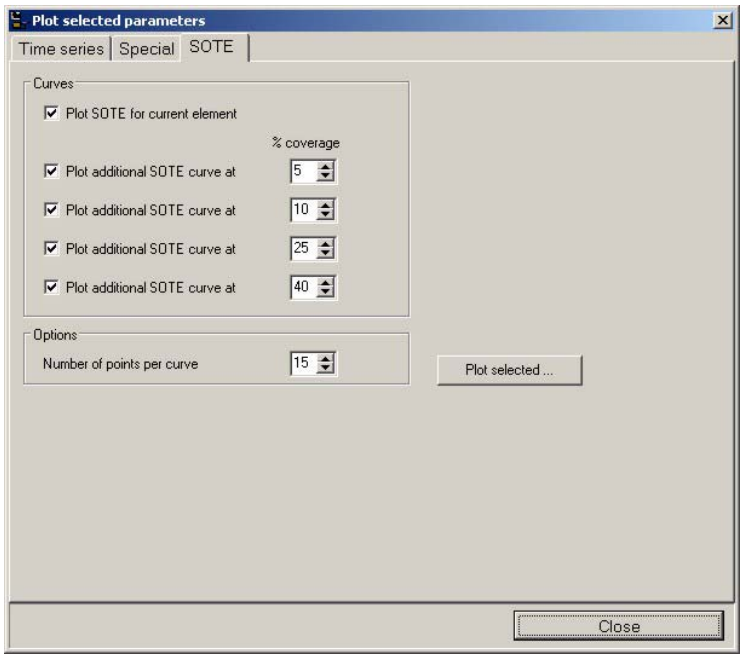
A SOTE (%) series is useful for comparing alternate aeration system designs. It allows you to quickly plot SOTE (%) as a function of airflow rate per diffuser for the current reactor diffuser coverage (the area of diffusers divided by the area of the tank). In addition, you may plot SOTE (%) as a function of airflow rate per diffuser for alternate diffuser coverage to compare a number of different design scenarios. Note that an **Album Template Page** with typical SOTE (%) numbers has been provided in the BioWin **Templates** directory.

Note: If an SOTE plot is generated for an SBR element that incorporates prezones, the plot information applies to the **main SBR zone only**. The information on the plot does not apply to the prezone(s).

Add a SOTE (%) Series

1. In the drawing board, right-click on the element that you wish to create a SOTE (%) series for.

2. From the resulting popup menu, select **Add to album**. Next, select **Chart…** from the resulting flyout. If you are presented with the choice of adding the series to an existing chart, click **Yes** or **No**.
3. On the **SOTE(%)** tab, select the cases you would like to plot SOTE for (i.e. for the current diffuser coverage and alternate coverage that you may specify using the **%coverage** spin edit boxes).



The SOTE(%) series dialog

4. You may change the number of pints that will be plotted in each series using the **Number of points per curve** setting. Increasing this value tends to result in smoother curves.
5. Click the **Plot selected…** button to plot the series. Click **Close** to close the dialog box. You may now open the album and look at your SOTE plot.

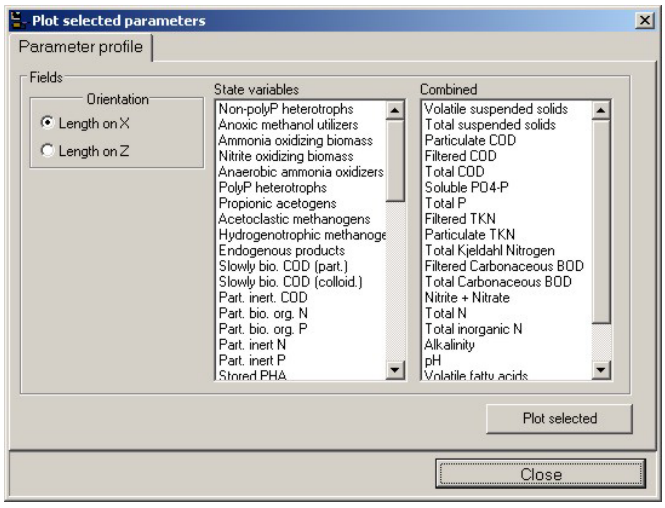
SBR Profile Series

A SBR profile series is useful for viewing how a parameter changes with depth over the length of a sequencing batch reactor. The resulting plot is a three-dimensional chart that shows SBR length along either the x-axis (horizontal axis) or the z-axis (axis going into the screen), depending on the option you choose. Concentration of the various parameters will be plotted on the y-axis (vertical axis), and the third axis will be assigned to SBR depth.

Add a SBR Profile Series

1. In the drawing board, right-click on the SBR element that you wish to create a SBR profile series for.
2. From the resulting popup menu, select **Add to album**. Next, select **Profile plot…** from the resulting flyout. If you are presented with the choice of adding the series to an existing chart, click **Yes** or **No**.

3. Using the dialog box shown below, select the **Orientation** of the profile plot by choosing to plot SBR length on the x-axis (**Length on X**) or SBR length on the z-axis (**Length on Z**).



The SBR profile series dialog box

4. From the **State variables** and **Combined** lists, choose the parameters you wish to plot and click the **Plot selected** button. Note that you can plot multiple parameters on one chart by repeating this step before clicking the **Close** button to finish.

Settling Tank Profile Series

A settling tank profile series is useful for viewing how a parameter changes over the depth of a settling tank element (or other separator-type elements such as grit removal tanks, dewatering units, and ideal primary clarifiers). There are two different types of profile series:

1. Current value
2. Time series (surface)

A **Current value** settling tank profile series shows the most recent (with respect to simulation start time) values of the chosen parameter(s) over the depth of the settling tank. During a dynamic simulation, the settling tank profile series will be updated and redrawn after each data interval. For a steady-state simulation, the settling tank profile series will represent the final steady-state values.

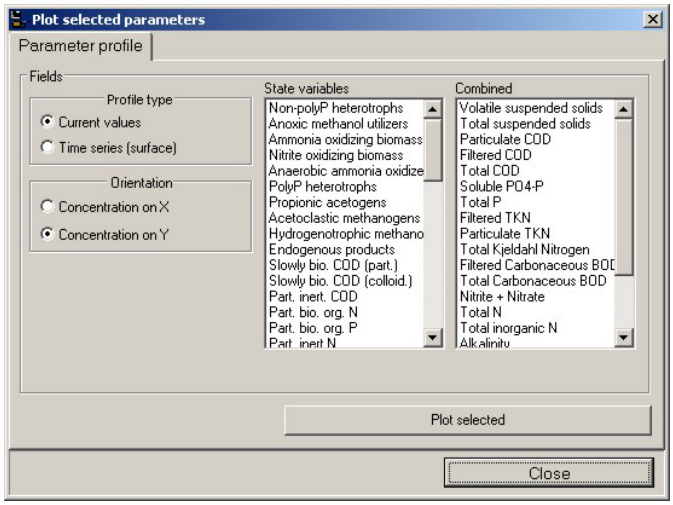
A **Time series (surface)** settling tank profile series shows the history of a parameter changing over the depth of a settling tank. This history is illustrated as a surface where the leading edge represents the most recent profile.

Add a Settling Tank Profile Series

1. In the drawing board, right-click on the settling tank element that you wish to create a profile series for.
2. From the resulting popup menu, select **Add to album**. Next, select **Profile plot…** from the resulting flyout. If you are presented with the choice of adding the series to an existing chart, click **Yes** or **No**.

3. Select the **Profile type**.

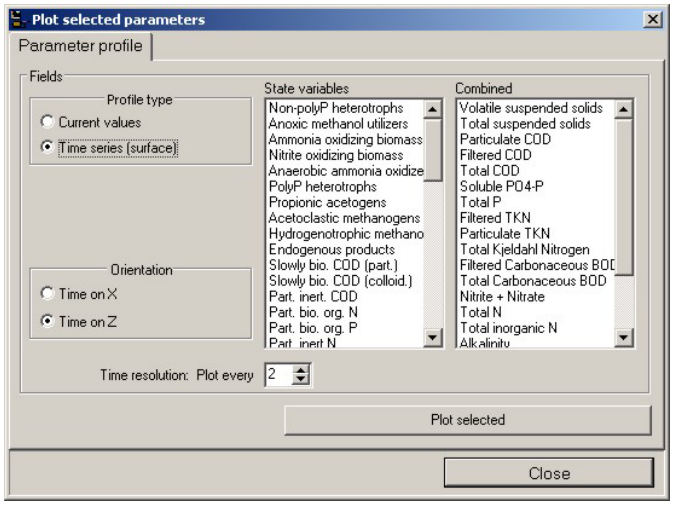
If you select a **Current value** settling tank profile:



The current value settler profile series dialog

1. From the **Orientation** radio button group, select whether you want to plot **Concentration on X**, or **Concentration on Y**. In making this choice you are specifying whether you want the concentration of the parameter you are plotting on the vertical (Y) or horizontal (X) axis. The settling tank depth will be plotted on the axis you do not select.
2. Select the parameter you want to plot the profile of from the **State variables** or **Combined** list boxes.
3. Click the **Plot selected** button.
4. From the **General series gallery**, choose the desired series style that you wish to apply. If you want your chart to have a three-dimensional appearance, ensure that the box labeled **3D** is checked and click **OK**.
5. Click the **Close** button to finish.

If you select a **Time series (surface)** settling tank profile:



The time series (surface) settler profile series dialog

1. From the **Orientation** radio button group, select whether you want to plot **Time on X**, or **Time on Z**. In making this choice you are specifying whether you want the time history of the parameter you are plotting to go along the horizontal (X) or Z (going into the screen) axis. Settling tank depth will be plotted on the axis you do not select, and for surface profiles concentration of the parameter you are plotting always is assigned to the vertical (Y) axis.
2. Select the parameter you want to plot the profile of from the **State variables** or **Combined** list boxes.
3. Set your **Plot resolution**. If you want a fine grid, decrease the value in the spin edit box. If you want a coarse grid, increase the value. Note that fine grids look better, but will increase demand on system resources.
4. Click the **Plot selected** button.
5. Click the **Close** button to finish.

Function Series

This section gives an overview of the general concept behind a function series, outlines the basic procedure for adding a function series to a chart, and gives detailed descriptions of the function series types available in BioWin.

A function is a series type available from the album that uses one or more existing series in a chart as its data source in order to perform a functional operation. For example, if you have two line series 'A' and 'B' and you apply the **ADD** function to those series, a third series 'C' will be generated which has the sum of series 'A' and 'B' values as its data source. Note that some of the functions available are able to use multiple series as inputs (e.g. the multiply function), while other functions may not have multiple series as inputs (e.g. the momentum function).

Once a function series has been added to a chart, it can be formatted and manipulated just like a regular series. There is one important property that distinguishes function series from non-function series. That property is **Period**.

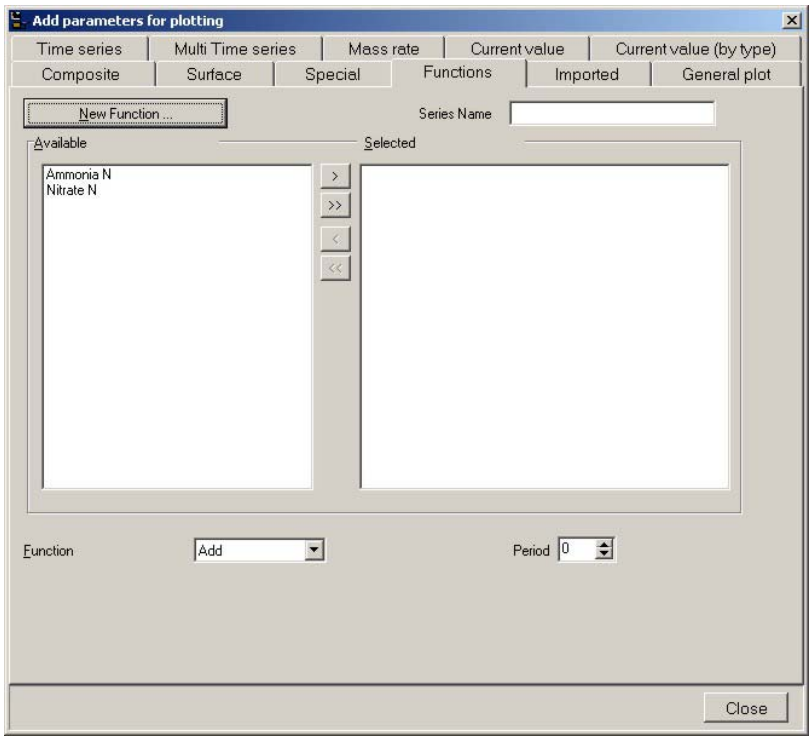
Period refers to the repetition of the chosen function. For example, say you have a series with data for an entire year. The average of all points for the year will yield one value (the year's average) that may be displayed as a flat line across the chart. The monthly average will calculate an average for each month and could be displayed as twelve different points across the chart of one year. This example shows that different results will be calculated depending upon the period defined.

Setting the period to a value of 0 means that all points in the series will be used and the function will only be calculated once. Setting the period to a value of 1 means that each point in the series will be used and the function will be calculated **n** times for **n** points. Setting the period to a value of 2 means that the function will be calculated on each successive group of two points so that the function will be calculated **n/2** times for **n** points. In general, setting the period to a value of **p** for **n** points means that the function will be calculated **n/p** times.

Add a Function Series

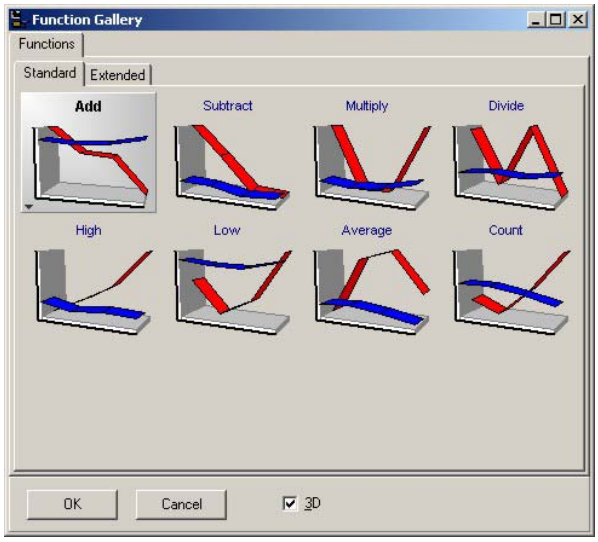
1. Right-click on the chart and click **Add Series** in the resulting pop-up menu.

2. On the **Functions** tab, click the **New Function…** button.



The function series dialog

3. Choose the function that you wish to apply from the **BioWin Functions Gallery**.



The function series gallery

4. From the **Available** list, select the series that you wish to use as the function series data source and move them to the **Selected** list in one of the following ways:

- You may move all of the series by clicking on the button marked with two right-pointing chevrons.
- You may move contiguous multiple series by clicking on the first desired series, and while holding down the **Shift** key, double-clicking the last desired series.
- You may move non-contiguous multiple series by holding the **Ctrl** key, clicking on the desired series, and double-clicking on the last desired series.
- You may move series one at a time by either double clicking on each series or clicking on each series and then clicking the button marked with one right-pointing chevron.

You may also use all of these techniques for removing series from the **Selected** list, using the buttons marked with left-pointing chevrons.

5. For certain function types, you may be presented with other options such as **Period**, **Fitted Curve Order**, **Weighted**, or **Weight %**. Set these to the appropriate value. For more information on these parameters, refer to the section regarding the specific function you have chosen.
6. If you wish you may enter a name for the function series in the **Series Name** text edit area.
7. When you are satisfied with your function series settings, click the **Close** button to finish. At any point before this final step you may change the type of function you are applying using the **Function** drop list box.

Add function

This function adds data from one or more series.

If only one series is defined for the add function, only values for that series will be used in function calculations. For example, if you have a series 'A' and you apply the add function to it with a period of 0, then the resulting function series will be a flat line representing the total of all the values for series 'A'.

If two or more series are defined for the add function, values for all the series defined will be used in function calculations. For example, if you have two series 'A' and 'B' and you apply the add function to them, the resulting function series will be a line with points corresponding to the sum of each pair of 'A' and 'B' values.

Average function

The average function takes the numerical average of values for one or more series.

$$\overline{Y} = \frac{\sum Y_i}{n}$$

where n = the number of points in the defined period

If only one series is defined for the average function, only values for that series will be used in function calculations. For example, if you have a series 'A' and you

apply the add function to it with a period of 0, then the resulting function series will be a flat line representing the average of all the values for series ‘A’.

If two or more series are defined for the average function, values for all the series defined will be used in function calculations. For example, if you have two series ‘A’ and ‘B’ and you apply the average function to them, then the resulting function series will be a line with points corresponding to the average of each pair of ‘A’ and ‘B’ values.

Divide function

The divide function divides data in one series by data from another series.

This function requires at least two input series. The second series defined for the function is the denominator; therefore the order in which the series are placed in the **Selected** list is important. For example, if you have two series ‘A’ and ‘B’ defined for the divide function in that order, the resulting function series will be a line with points corresponding to the division of each pair of (‘A’ / ‘B’) values.

If you add more than two series then the first series will be divided by the second, then that result is divided by the third, and so on.

Count function

The count function returns the number of points in an input data series. This function does not apply to multiple input series. If more than one series in placed in the **Selected** list, only the first will be used. Note also that since the function needs to look at each point in the input series, no choice is given as to what the period is set to.

For example, say that a series ‘A’ with 100 data points is the selected series for the count function. The resulting function series will be a flat line across the chart originating at 100 on the vertical axis.

Cumulative function

The cumulative function adds each point of an input data series in succession to give a total cumulative value for the input series values. Since this function needs to look at each data point in the input series, no choice is given as to what the period is set to.

For example, say a series ‘A’ is selected for the cumulative function. The resulting cumulative function series will be a line originating at the same location as the first point in series ‘A’ with each successive point representing the “running total” of series ‘A’.

If more than one series is selected, then each point of the cumulative function represents the sum of the “running totals” for each series.

Curve fit function

The curve fit function performs a polynomial Gaussian calculation on the input series data and draws a smooth curve over the input series points. Note that you are not asked to define a period for the curve fitting function, as the function requires that all points be used in order to perform a proper fit. Also, this function does not apply to multiple series.

The "order" of a polynomial may also be referred to as its "degree".

You must specify the **Fitted Curve Order** that you wish to apply to your data. This is the order of the polynomial that will be fit to the data. For example, entering a value of 2 for this parameter means that a quadratic polynomial will be fit to the data. Entering 3 will attempt to fit a cubic polynomial, and so on. Note that the maximum curve order calculated by BioWin is 15.

Exponential average function

This function calculates an exponentially weighted moving average according to the following formula:

$$\overline{Y_i} = \overline{Y}_{i-1} \cdot (1 - w) + Y_i \cdot w$$

where $\overline{Y_i}$ = the average at series point i

$\overline{Y}_{i-1}$ = the average calculated using the (i - 1) points

w = the specified weighting factor

Y_i = the current Y value of the series

High weighting factors give more weight to recent series values, and low weight factors give more weight to historical series points.

High function

The high function picks out the maximum value of one or more series.

If two or more series are defined for the high function, values for all the series defined will be used in function calculations at each period point. For example, if you have two series 'A' and 'B' and you apply the high function to them, then the resulting function series will be a line with points corresponding to the maximum of each pair of 'A' and 'B' values.

Low function

The low function picks out the minimum value of one or more series.

If two or more series are defined for the low function, values for all the series defined will be used in function calculations at each period point. For example, if you have two series 'A' and 'B' and you apply the low function to them, then the resulting function series will be a line with points corresponding to the minimum of each pair of 'A' and 'B' values.

Momentum function

The momentum function subtracts the first value from the last value over a defined period. This function only applies to a single input series. You may find that this function is especially useful for steady state analysis. When a system is at steady state, the momentum function over a defined period (e.g. 1 day) for a compound (e.g. TSS) should be close to zero.

For example, if you are running a simulation at an initial Solids Retention Time (SRT) and you make a step change to the SRT by changing your wastage rate, the system will need about 3-4 SRTs to return to steady state. You can use the momentum function to indicate when your system has returned to steady state.

Initially after the change in SRT, you will notice the momentum function move away from zero. As the system returns to steady state, you should see the momentum function return to zero.

Moving average function

A *p* period moving average function provides a calculated average value for the current and preceding (*p*-1) points according to the following formula:

$$\overline{Y_i} = \frac{1}{p}\sum_{j=1}^{p} Y_{i-j+1}$$

where $\overline{Y_i}$ = the moving average of the Y values at series point i

p = the specified moving average period

Y_i = the current Y value of the series

You also are given the option to calculate a weighted moving average, according to the following formula:

$$\overline{Y_i} = \frac{\sum_{j=1}^{p} Y_{i-j+1} \cdot X_{i-j+1}}{\sum_{j=1}^{p} X_{i-j+1}}$$

where X_i = the current X value of the series

p = the specified moving average period

Y_i = the current Y value of the series

In contrast to the ordinary moving average where all of the preceding data points are weighted equally, for the weighted average the most recent points are weighted more heavily and on the basis of the value of X rather than on the number of points. This may be useful for calculating a moving average on imported data series in which the X values are not evenly spaced.

Multiply function

The multiply function multiplies data for as many series as you like.

If two or more series are defined for the multiply function, values for all the series defined will be used in function calculations. For example, if you have two series 'A' and 'B' and you apply the multiply function to them, then the resulting function series will be a line with points corresponding to the product of each pair of 'A' and 'B' values.

Subtract function

The subtract function subtracts data in one series from another series.

This function requires two input series. The second series defined for the function will be subtracted from the first, so the order in which the source series are placed in the **Selected** list is important. For example, if you have two series 'A' and 'B' defined for the subtract function in that order, the resulting function series will be a line with points corresponding to the subtraction of each pair of 'A' and 'B' values.

Trend function

The trend function fits a best fit trend line to data points. This function does not apply to multiple series, so if more than one series is placed in the **Selected** list, a trend line will be fit to the first in the list. Also, since this function requires that each point in the input data series be used, no choice is given as to what the period may be.

Chart Formatting Procedures

Chart Formatting Note

One of BioWin's helpful charting features is that changes you make to a chart are displayed "on the fly" as you make them. This is an excellent way to learn the functionality of the chart editing/formatting controls since you immediately see the results of clicking various buttons, spin edit boxes, and other dialog box controls in the background. The associations between dialog controls and their resultant changes to the chart are learned much more quickly than if you have to exit the dialog or manually apply the changes before you see their effects.

In many instances entering a value of "0" in a chart formatting field means that BioWin will resort to a default value for the selected property (eg. the increment of an axis scale).

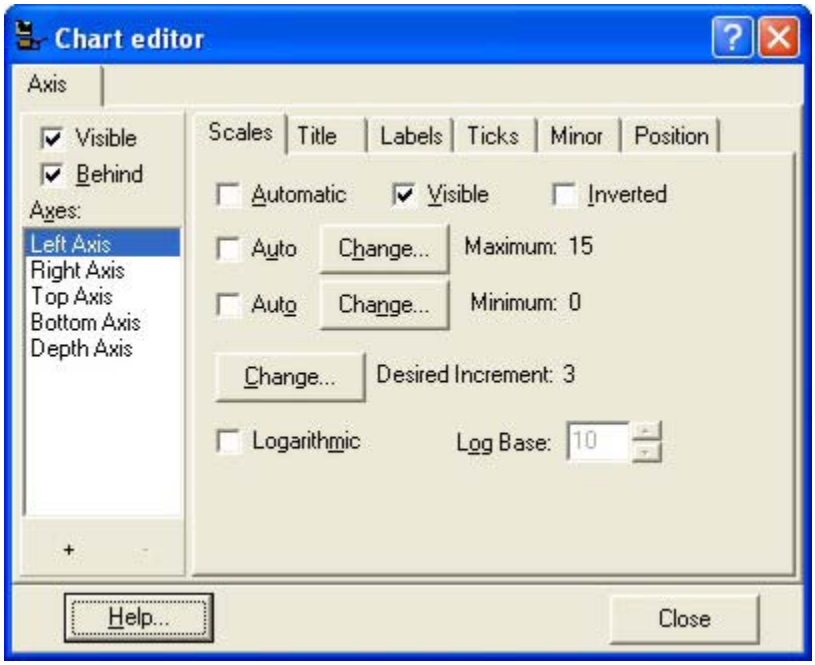
Chart Axis Procedures

This section outlines the basic procedure you need to know in order to change and manipulate chart axis properties, and also gives a number of specific related procedures. The section is structured such that the order of its sub-topics follows the order of the sub-tabs on the **Edit Axes** main tab.

The key is to understand the use of the **Axes** list box. In general, most axis manipulations are done via the following procedure:

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis (i.e. **Left**, **Right**, **Top**, **Bottom**) that you want to change with the **Axes** list.
3. Select one of the four sub-tabs to change general axis properties such as **Scale**, **Titles**, etc.
4. Make the desired specific changes to the axis you have specified using the options on the sub-tabs.

Next, some procedures that may be accessed directly from the **Edit Axis** main tab will be outlined.



Axis editor tab and sub-tabs

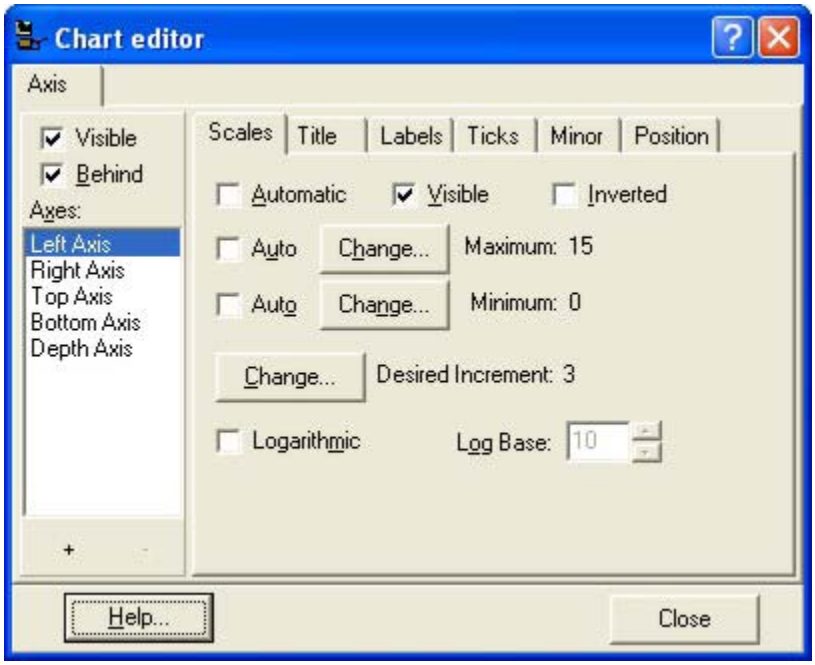
Remove a Chart Axis

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. In the **Axes** list, select the axis you want to remove.
3. Un-check the box labeled **Visible**.
4. Click the **Close** button to finish.

Note: To remove another axis, repeat steps 1-3.

Axis Scale Procedures

This section contains procedures that can be executed from the **Scale** sub-tab of the **Edit Axes** tab. The topics all are related to controlling the numerical properties of the chart axes such as scale increment, scale maximum and minimums, and the type of axis (i.e. logarithmic, inverted).



The axis scale editor sub-tab

Set Axis Scale Maximum and Minimum Values

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to set scale maximum and/or minimum values for using the **Axes** list.
3. If you want BioWin to set the scale maximum and minimum automatically, check the box labeled **Automatic** and go to step 8. If you want to manually set one or both of the values, go to Step 4.
4. Un-check the box labeled **Automatic**.
5. Uncheck **Auto** button for the value (i.e. maximum or minimum) you wish to manually set and click the **Change** button.
6. Enter your desired value in the resulting dialog box and click **OK**.
   - If you have set the axis to Date/Time Format, then the dialog box will allow you to set the desired maximum or minimum in date/time formats.
7. If you would like to set the remaining value (i.e. maximum or minimum) manually, repeat Step 5. To have BioWin automatically set the remaining value, check the box labeled **Auto** for that value.
8. Click the **Close** button to finish.

Note: If you do not see the series you expect to when you first look at a chart, check the box labeled **Automatic** to ensure that all series are contained within the axes boundaries.

Set Axis Scale Increment

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.

2. Select the axis you wish to set scale increment for using the **Axes** list.
3. Click the **Change...** button.
4. Enter your desired increment in the resulting dialog box and click **OK**.
    - If you have set the axis to Date/Time Format, then the dialog box will allow you to set the desired increment in time units.
5. Click the **Close** button to finish.

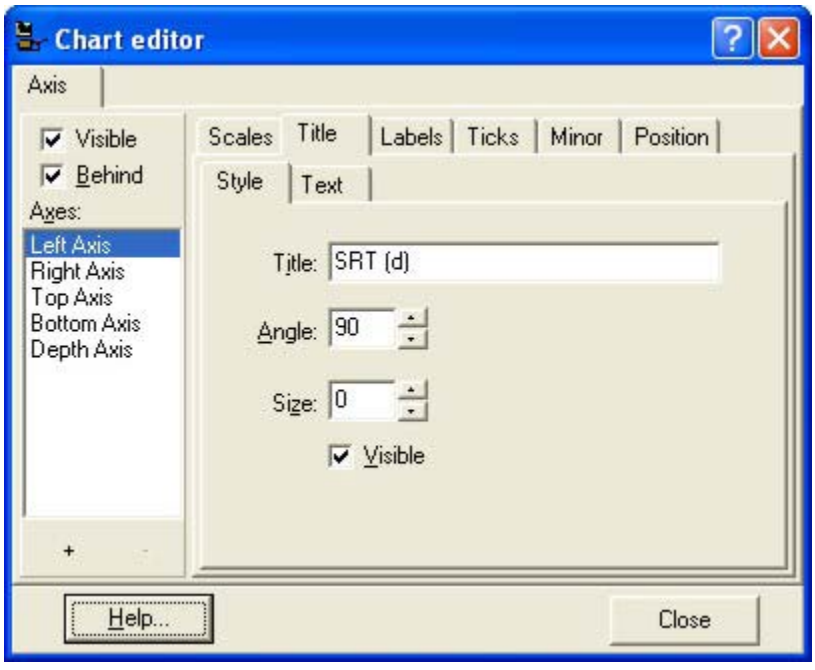
Note : Entering a value of "0" in the Axis Increment field will result in BioWin determining the axis increment automatically.

Set Axis Scale Type

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to set the scale type for using the **Axes** list.
    - If you want the selected axis to have a logarithmic scale, check the box labeled **Logarithmic**.
    - If you want the axis scale inverted (i.e. minimum value at the top of a vertical axis), check the box labeled **Inverted**.
3. Click the **Close** button to finish.

Axis Title Procedures

This section contains procedures that can be executed from the **Title** sub-tab of the **Edit Axes** tab. The topics covered all are related to creating and formatting chart axis titles, including adding a title, changing the axis title font, changing the axis title area size, and changing the axis title angle.



The axis title editor sub-tab

Add an Axis Title

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to add a title to using the **Axes** list.
3. Click the **Title** sub-tab.
4. In the text edit area labeled **Title**, enter the text for your title.
5. Click the **Close** button to finish.

Change the Axis Title Area Size

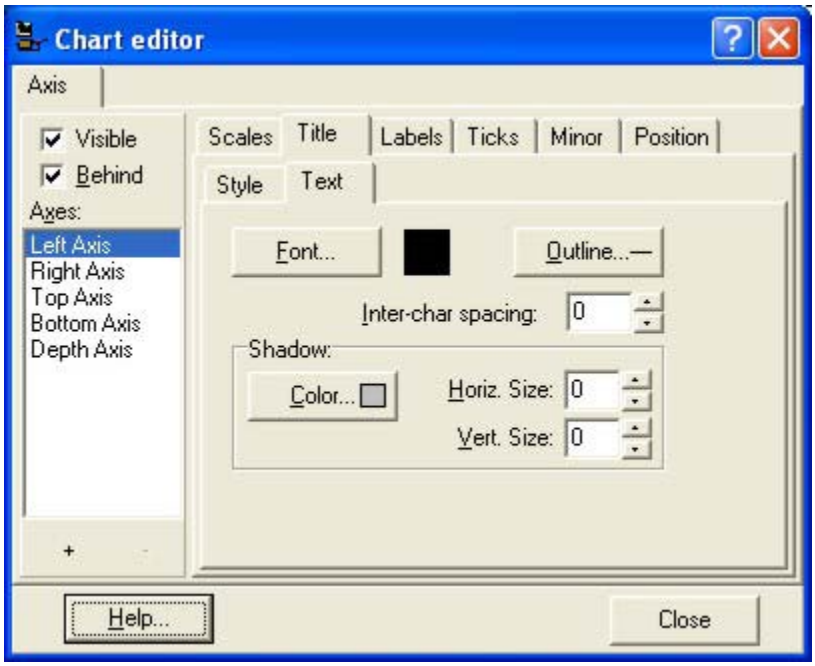
1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to change the title area size for using the **Axes** list.
3. Click the **Title** sub-tab.
4. If you have not already done so, enter the text for your title in the text edit area labeled **Title**.
5. Click the spin edit box labeled **Size** to change the title area size by 1-point increments, or enter a value using the keyboard.
6. Click the **Close** button to finish.

Note : Entering a value of "0" in the Axis Title Area Size field will result in BioWin determining the size automatically.

Change the Axis Title Angle

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to change the title angle for using the **Axes** list.
3. Click the **Title** sub-tab.
4. If you have not already done so, enter the text for your title in the text edit area labeled **Title**.
5. Click the spin edit box labeled **Angle** to change the title angle by 90-degree increments, or enter a value using the keyboard.
6. Click the **Close** button to finish.

Change the Axis Title Font



The axis title text editor sub-tab

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to change the title font for using the **Axes** list.
3. Click the **Title** sub-tab.
4. If you have not already done so, enter the text for your title in the text edit area labeled **Title**.
5. Click the **Text** sub-tab to display the font properties.
6. Click the **Font…** button to display the Font Properties dialog box and select the **Font** (e.g. Arial, Times New Roman, etc), the **Font Style** (e.g. Italic, Bold, etc.), **Size**, and **Color** until the **Sample Text** has the appearance you want.
7. Click the **OK** button to close the Font Properties dialog box.
8. Click the **Color** button in the **Shadow…** group to display the Color dialog box and select from the **Basic Colors** to add a shadow to your text if desired.
9. Click the **OK** button to close the Color dialog box.
10. Click the spin edit boxes to adjust the horizontal and vertical aspects of the shadow or enter a value from the keyboard.
11. Click the **Inter-char spacing** spin edit box to adjust the character spacing or enter a value from the keyboard.
12. Click the **Close** button to finish.

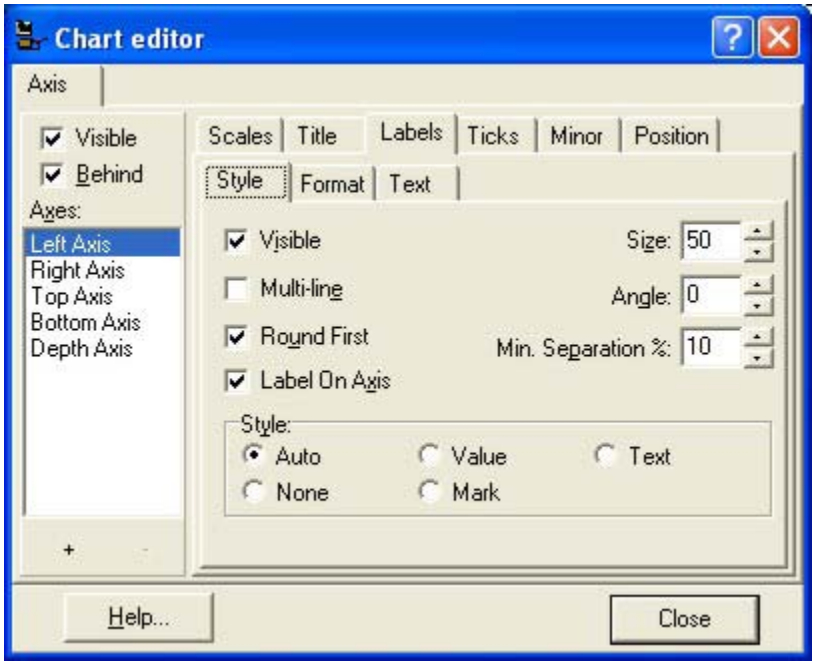
Change the Axis Title Border

The axis title border editor dialog box

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to change the title font for using the **Axes** list.
3. Click the **Title** sub-tab.
4. If you have not already done so, enter the text for your title in the text edit area labeled **Title**.
5. Click the **Text** sub-tab to display the font properties.
6. Click the **Outline…** button to display the Border Editor dialog box and select the **Style**, **Color** and **Width** of border you require around the title.
7. Click the **OK** button to close the Border Editor dialog box.
8. Click the **Close** button to finish.

Axis Label Procedures

This section contains procedures that can be executed from the **Labels** sub-tab of the **Edit Axes** tab. The topics covered include procedures for changing the axis label font, position, and style.



The tick labels editor sub-tab

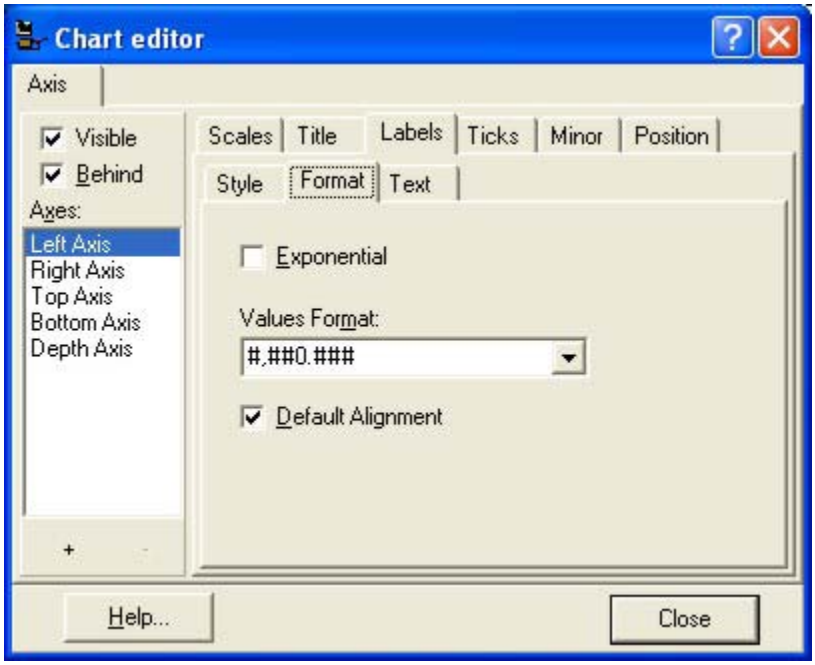
Change the Axis Label Options

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to change the label font for using the **Axes** list.
3. Click the **Labels** sub-tab.
4. Ensure that the box labeled **Visible** is checked.
5. Click the spin edit box labeled **Size** to change the amount of space between the title and the axes labels by 1-point increments, or enter a value using the keyboard.
7. Click the spin edit box labeled **Angle** to change the label angle by 90-degree increments, or enter a value using the keyboard.
8. Click the spin edit box labeled **Min Separation %** to change the space between axis labels in 10 percent increments, or enter a value using the keyboard.
9. Check the box labeled **Multi-line** to word wrap your axis label.
10. Check the box labeled **Round First** to round multi-decimal labels to one decimal point. Note this option only works with certain styles (eg. Auto and Value).
11. Check the box labeled **Label on Axis** to include or exclude the first and last axis labels.
12. Click the **Close** button to finish.

Change the Axis Label Style

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to change the label font for using the **Axes** list.
3. Click the **Labels** sub-tab.
4. Ensure that the box labeled **Visible** is checked.
5. Use the radio buttons in the area labeled **Style** to adjust the display of the axis values (eg. Text, Label, Value).
6. Click the **Close** button to finish.

Change the Axis Label Format



The axis label format dialog box

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to change the label font for using the **Axes** list.
3. Click the **Labels** sub-tab.
4. Ensure that the box labeled **Visible** is checked.
5. Click the **Format** sub-tab.
6. Use the drop-down list labeled **Values Format** to adjust the display of the axis values or see the sections **BioWin Number Formats** and **BioWin Date / Time Formats**.
7. Click the **Close** button to finish.

BioWin Number Formats

This section contains an overview of the various number formats that are available in BioWin. Although this information is applicable elsewhere, it has been included here since it will most likely be needed in relation to chart axis labeling.

Specifier	Description
0	Digit placeholder. If the value being formatted has a digit in the position where the '0' appears in the format string, then that digit is copied to the output string. Otherwise, a '0' is stored in that position in the output string.
#	Digit placeholder. If the value being formatted has a digit in the position where the '#' appears in the format string, then that digit is copied to the output string. Otherwise nothing is stored in that position in the output string.
.	Decimal point. The first '.' character in the format string determines the location of the decimal separator in the formatted value; any additional '.' characters are ignored. The actual character used as a decimal separator in the output string is specified in the Number Format of the International section of the Windows Control Panel.
,	Thousand separator. If the format string contains one or more ',' characters, the output will have thousand separators inserted between each group of three digits to the left of the decimal point. The placement and number of ',' characters in the format string does not affect the output, except to indicate that thousand separators are wanted. The actual character used as a thousand separator in the output string is specified in the Number Format of the International section of the Windows Control Panel.
E+	Scientific notation. If any of the strings 'E+', 'E-', 'e+', or 'e-' are contained in the format string, the number is formatted using scientific notation. A group of up to four '0' characters can immediately follow the 'E+', 'E-', 'e+', or 'e-' to determine the minimum number of digits in the exponent. The 'E+' or 'e+' formats cause a plus sign to be output for positive exponents and a minus sign to be output for negative exponents. The 'E-' and 'e-' formats output a sign character only for negative exponents.
'xx'/"xx"	Characters enclosed in single or double quotes are output as-is, and do not affect formatting.
;	Separates sections for positive, negative, and zero numbers in the format string.

The locations of the leftmost '0' before the decimal point in the format string and the rightmost '0' after the decimal point in the format string determine the range of digits that are always present in the output string.

The number being formatted is always rounded to as many decimal places as there are digit placeholders ('0' or '#') to the right of the decimal point. If the format string contains no decimal point, the value being formatted is rounded to the nearest whole number.

If the number being formatted has more digits to the left of the decimal separator that there are digit placeholders to the left of the '.' character in the format string, the extra digits are output before the first digit placeholder.

To allow different formats for positive, negative, and zero values, the format string can contain between one and three sections separated by semicolons:

1. One section: The format string applies to all values;

2. Two sections: The first section applies to positive values and zeros, and the second section applies to negative values;
3. Three sections: The first section applies to positive values, the second applies to negative values, and the third section applies to zeros.

If the section for negative values or the section for zero values is empty, that is there is nothing between the semicolons that delimit the section, the section for the positive values is used instead.

If the section for positive values is empty, or if the entire format string is empty, the value is formatted using general floating-point formatting with 15 significant digits. General floating-point formatting is also used if the value has more than 18 digits to the left of the decimal point and the format string does not specify scientific notation.

The following table shows some sample formats and the results produced when the formats are applied to different values:

Format String	1234	-1234	0.5	0
None Specified	1234	-1234	0.5	0
0	1234	-1234	1	0
0.00	1234.00	-1234.00	0.50	0.00
#.##	1234	-1234	.5	
#,##0.00	1,234.00	-1,234.00	0.50	0.00
#,##0.00;(#,##0.00)	1,234.00	(1,234.00)	0.50	0.00
#,##0.00;;Zero	1,234.00	-1,234.00	0.50	Zero
0.000E+00	1.234E+03	-1.234E+03	5.000E-01	0.000E+00
#.###E-0	1.234E3	-1.234E3	5E-1	0E0

BioWin Date / Time Formats

This section contains an overview of the various date/time formats that are available in BioWin. Although this information is applicable elsewhere, it has been included here since it will most likely be needed in relation to chart axis labeling.

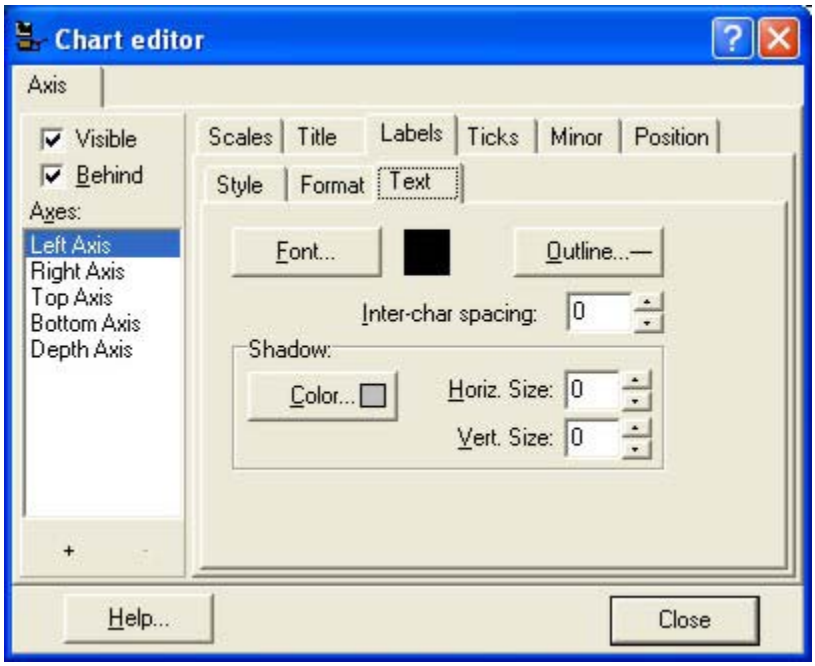
Specifier	Description
C	Displays the date followed by the time.
D	Displays the day as a number without a leading zero (1-31).
dd	Displays the day as a number without a leading zero (01-31).
ddd	Displays the day as an abbreviation (Sat-Sun).
dddd	Displays the day as a full name (Saturday-Sunday).
ddddd	Displays the date in a short format.
dddddd	Displays the date in a long format.
M	Displays the month as a number without a leading zero (1-12). If the ‘m’ specifier immediately follows an ‘h’ or ‘hh’ specifier, the minute rather than the month is displayed.
mm	Displays the month as a number with a leading zero (01-12). If the ‘mm’ specifier immediately follows an ‘h’ or ‘hh’ specifier, the minute rather than the month is displayed.
mmm	Displays the month as an abbreviation (Jan-Dec).
mmmm	Displays the month as a full name (January-December).
yy	Displays the year as a two-digit number (00-99).
yyyy	Displays the year as a four-digit number (0000-9999).

H	Displays the hour as a number without a leading zero (0-23).
hh	Displays the hour as a number with a leading zero (00-23).
N	Displays the minute as a number without a leading zero (0-59).
nn	Displays the minute as a number with a leading zero (00-59).
S	Displays the second as a number without a leading zero (0-59).
ss	Displays the second as a number with a leading zero (00-59).
T	Displays the time using a short format.
Tt	Displays the time using a long format.
am/pm	Uses the 12-hour clock for the preceding ‘h’ or ‘hh’ specifier, and displays ‘am’ for any hour before noon, and ‘pm’ for any hour after noon. The am/pm specifier can use lower, upper, or mixed case, and the result is displayed accordingly.
a/p	Uses the 12-hour clock for the preceding ‘h’ or ‘hh’ specifier, and displays ‘a’ for any hour before noon, and ‘p’ for any hour after noon.
Ampm	Uses the 12-hour clock for the preceding ‘h’ or ‘hh’ specifier, and displays a default am string for any hour before noon, and a default pm string for any hour after noon.
/	Displays the date separator character.
:	Displays the time separator character.
'xx'/"xx"	Characters enclosed in single or double quotes are displayed as-is, and do not affect formatting.

Format specifiers may be written in upper case as well as lower case letters - both produce the same result.

If the string given by the format parameter is empty, the date and time is formatted as if a ‘c’ format specifier had been given.

Change the Axis Label Font



The axis label text editor dialog box

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to change the label font for using the **Axes** list
3. Click the **Labels** sub-tab.
4. Ensure that the box labeled **Visible** is checked.
5. Click the **Text** sub-tab to display the font properties.
6. Click the **Font…** button to display the Font Properties dialog box and select the **Font** (e.g. Arial, Times New Roman, etc), the **Font Style** (e.g. Italic, Bold, etc.), **Size**, and **Color** until the **Sample Text** has the appearance you want.
7. Click the **OK** button to close the Font Properties dialog box.
8. Click the **Color** button in the **Shadow…** group to display the Color dialog box and select from the **Basic Colors** to add a shadow to your text if desired.
9. Click the **OK** button to close the Color dialog box.
10. Click the spin edit boxes to adjust the horizontal and vertical aspects of the shadow or enter a value from the keyboard.
11. Click the **Inter-char spacing** spin edit box to adjust the character spacing or enter a value from the keyboard.
12. Click the **Close** button to finish.

Change the Axis Label Border

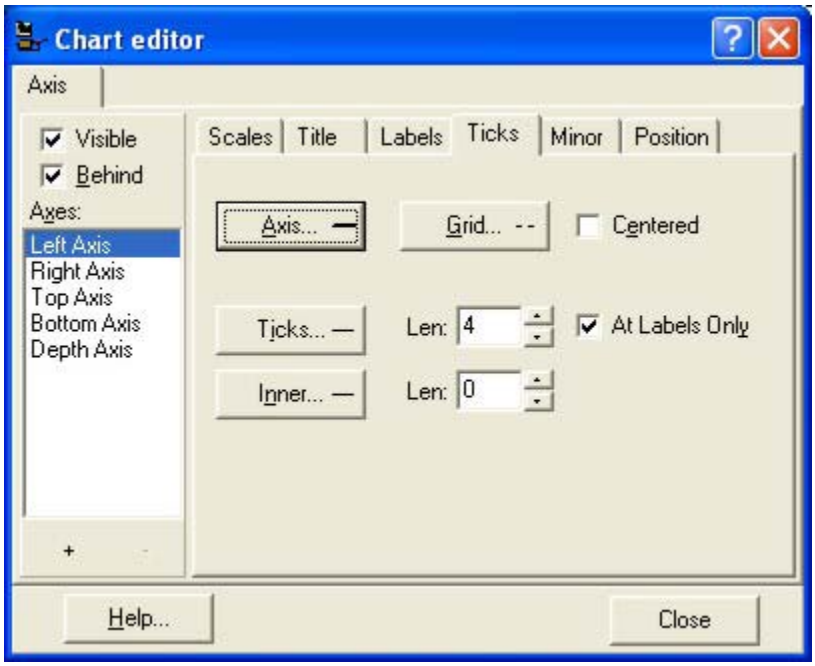
The axis title border editor dialog box

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to change the title font for using the **Axes** list.
3. Click the **Label** sub-tab.
4. If you have not already done so, enter the text for your title in the text edit area labeled **Axis Title**.
5. Click the **Text** sub-tab to display the font properties.
6. Click the **Outline…** button to display the Border Editor dialog box and select the **Style**, **Color** and **Width** of border you require around the title.
7. Click the **OK** button to close the Border Editor dialog box.

8. Click the **Close** button to finish.

Tick Formatting Procedures

This section contains procedures that can be executed from the **Tick Formats** sub-tab of the **Edit Axes** tab. The topics covered include procedures for specifying the appearance of the chart axes, grid, and axes ticks.



The tick formatting editor sub-tab

Change the Chart Axis Formatting

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to change the formatting for using the **Axes** list.
3. Click the **Ticks** sub-tab.
4. Click the **Axis...** button to open the border color editor.
5. Make sure that the box labeled **Visible** is checked.
6. Select the axis width you want from the **Width** spin edit box.
7. To change the axis color, select the **Color...** button to open the color selection dialog box.
8. Click on the axis color you want and then click **OK** to close the color selection dialog box.
9. Click **OK** to close the border editor.
10. Click the **Close** button to finish.

Change the Chart Major Grid Formatting

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to change the formatting for using the **Axes** list.
3. Click the **Ticks** sub-tab.
4. Click the **Grid...** button to open the border editor.
5. Make sure that the box labeled **Visible** is checked.
6. Select the grid style you want from the **Style** drop-down list.
7. Select the grid width you want from the **Width** spin edit box.
8. To change the grid color, select the **Color...** button to open the color selection dialog box.
9. Click on the grid color you want and then click **OK** to close the color selection dialog box.
10. Click **OK** to close the border editor.
11. Click the **Close** button to finish.

Change the Axis Major Tick Formatting

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to change the formatting for using the **Axes** list.
3. Click the **Ticks** sub-tab.
4. Click the **Ticks...** button to open the border editor.
5. Make sure that the box labeled **Visible** is checked.
6. Select the major tick style you want from the **Style** drop-down list.
7. Select the major tick width you want from the **Width** spin edit box.
8. To change the major tick color, select the **Color...** button to open the color selection dialog box.
9. Click on the major tick color you want and then click **OK** to close the color selection dialog box.
10. Click **OK** to close the border color editor.
11. If you only want the major ticks where labels are, ensure that the box labeled **At Labels Only** is checked.
12. If you want to increase the size of the major ticks, use the **Len** spin edit box.
13. Click the **Close** button to finish.

Change the Axis Inner Tick Formatting

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.

2. Select the axis you wish to change the formatting for using the **Axes** list.
3. Click the **Ticks** sub-tab.
4. Click the **Inner...** button to open the border editor.
5. Make sure that the box labeled **Visible** is checked.
6. Select the major tick style you want from the **Style** drop-down list.
7. Select the major tick width you want from the **Width** spin edit box.
8. To change the major tick color, select the **Color...** button to open the color selection dialog box.
9. Click on the major tick color you want and then click **OK** to close the color selection dialog box.
10. Click **OK** to close the border editor.
11. If you only want the major ticks where labels are, ensure that the box labeled **At Labels Only** is checked.
12. If you want to increase the size of the major ticks, use the **Len** spin edit box.
13. Click the **Close** button to finish.

Change the Axis Minor Tick Formatting

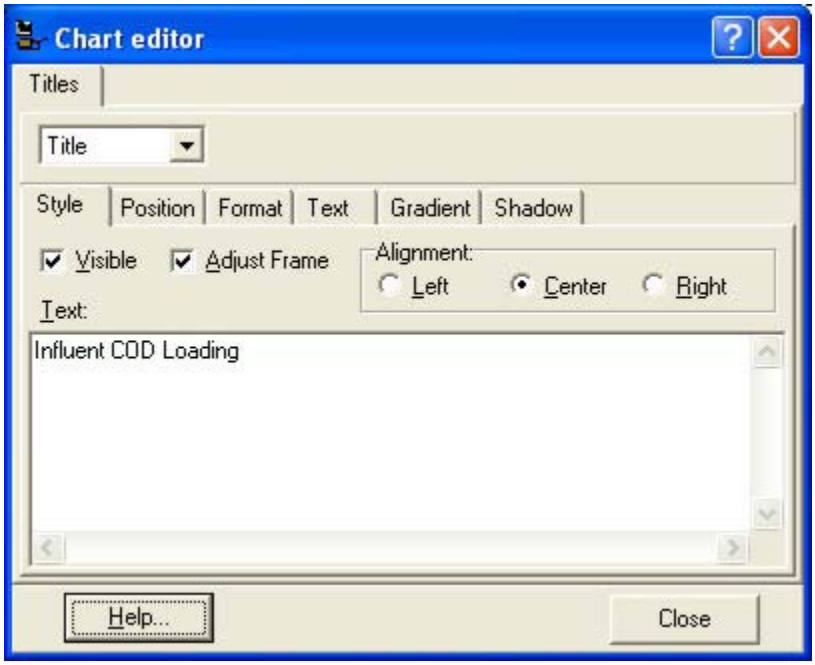
1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to change the formatting for using the **Axes** list.
3. Click the **Minor** sub-tab.
4. Click the **Ticks...** button to open the border editor.
5. Make sure that the box labeled **Visible** is checked.
6. Select the minor tick style you want from the **Style** drop-down list.
7. Select the minor tick width you want from the **Width** spin edit box.
8. To change the minor tick color, select the **Color...** button to open the color selection dialog box.
9. Click on the minor tick color you want and then click **OK** to close the color selection dialog box.
10. Click **OK** to close the border editor.
11. If you only want the minor ticks where labels are, ensure that the box labeled **At Labels Only** is checked.
12. If you want to increase the size of the minor ticks, use the **Length** spin edit box.
13. If you want to increase the number of minor ticks, use the **Count** spin edit box.
14. Click the **Close** button to finish.

Change the Chart Minor Grid Formatting

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Axes**.
2. Select the axis you wish to change the formatting for using the **Axes** list.
3. Click the **Minor** sub-tab.
4. Click the **Grid...** button to open the border editor.
5. Make sure that the box labeled **Visible** is checked.
6. Select the grid style you want from the **Style** drop-down list.
7. Select the grid width you want from the **Width** spin edit box.
8. To change the grid color, select the **Color...** button to open the color selection dialog box.
9. Click on the grid color you want and then click **OK** to close the color selection dialog box.
10. Click **OK** to close the border editor.
11. Click the **Close** button to finish.

Chart Title Procedures

This section contains procedures that can be executed from the **Edit Title** tab. Note that in BioWin charting terminology, a chart title may be either a **Title** at the top of the chart, or Footer at the bottom of the chart. It is possible to have both on the same chart, as well as a **Subtitle** and **Subfooter**. This section includes procedures for adding a chart title, changing a title's font and alignment, adding a frame to a title, and adding a fill pattern, gradient or shadow to the title frame.



The chart title style editor tab

Add a Chart Title

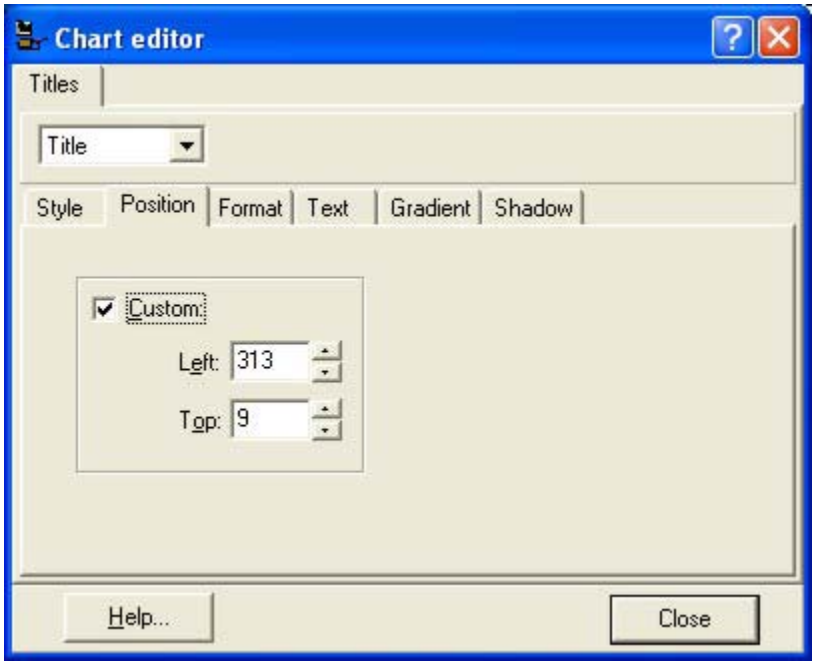
1. Right-click on the chart to open the **Edit** menu, and then click **Edit Titles**.
2. If you want the title above the chart, select **Title** in the drop list box. If you want the title below the chart, select **Foot**.
3. Ensure that the box labeled **Visible** is checked.
4. In the **Text** edit area; change the default text to the title you desire.
5. Click the **Close** button to finish.

Note: If no chart title is desired, un-check the box labeled **Visible**. If titles above and below the chart are desired, perform the above procedure twice, i.e. once for the **Title** and once for the **Foot**. A **Subtitle** and **Subfooter** can be added in the same way.

Change the Chart Title Alignment

1. Specify which title you want to change in the drop-down list box.
2. In the **Alignment** radio button group select **Left**, **Center**, or **Right**.
3. Click the **Close** button to finish.

Change the Chart Title Position

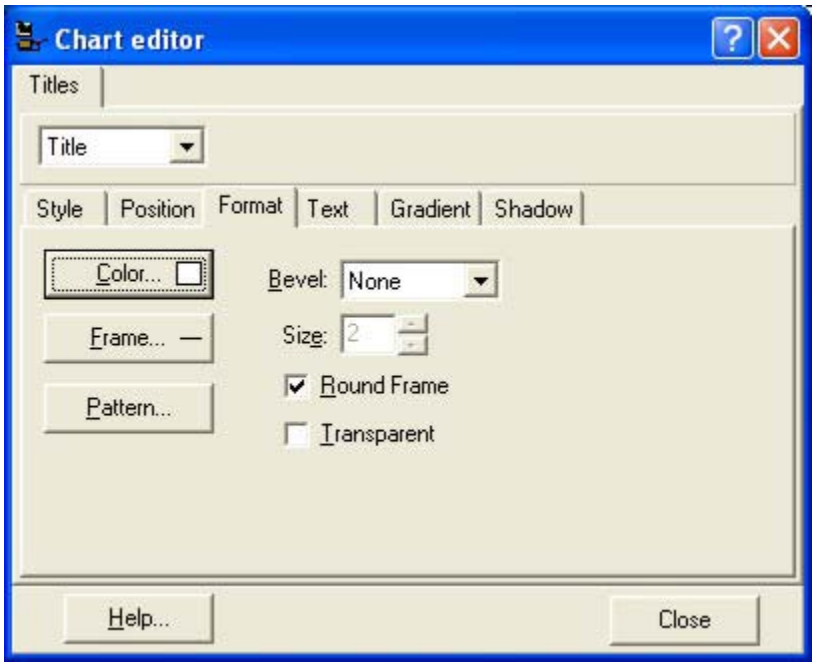


The chart title position editor tab

1. Specify which title you want to change in the drop-down list box.
2. Select the **Position** sub-tab.
3. Check the **Custom** box.

4. Adjust the **Left** and **Top** measurements by clicking the spin edit boxes or type in a value from the keyboard.
5. Click the **Close** button to finish.

Add a Frame to the Chart Title



The chart title format editor tab

1. Specify which title you want to change in the drop-down list box.
2. Select the **Format** sub-tab.
3. Make sure that the **Transparent** check box is unchecked.
4. Click the **Frame...** button to open the border editor.
5. Make sure that the box labeled **Visible** is checked.
6. Select the frame style you want from the **Style** drop-down list.
7. Select the frame width you want from the **Width** spin edit box.
8. To change the frame color, select the **Color...** button to open the color selection dialog box.
9. Click on the frame color you want and then click **OK** to close the color selection dialog box.
10. Click **OK** to close the border editor.
11. Click the **Close** button to finish.

Add a Fill Pattern and Color to a Chart Title Frame

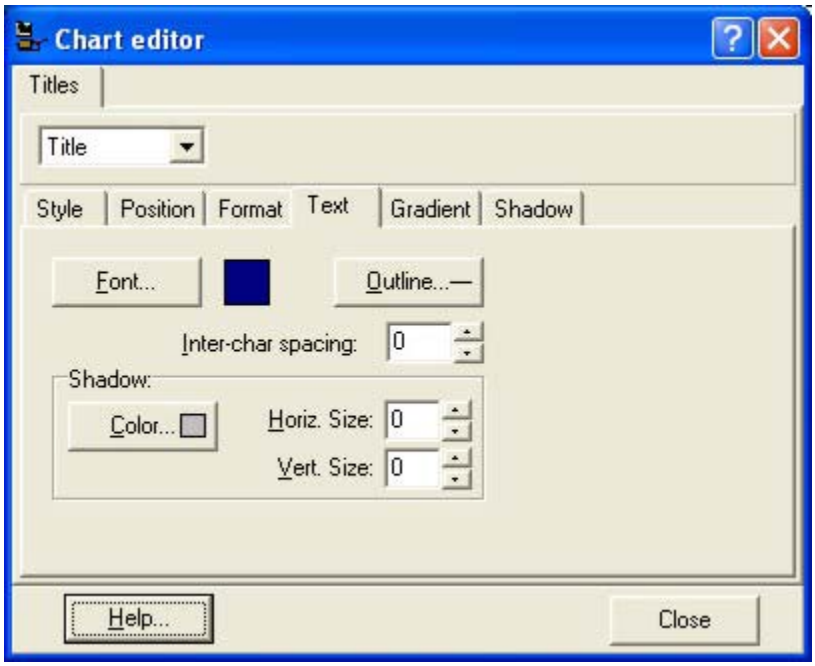
The pattern color editor dialog box

1. Specify which title you want to change in the drop-down list box.
2. Select the **Format** sub-tab.
3. Click the **Pattern...** button to open the fill pattern editor.
4. Select the fill style you want from the list box.
5. To change the fill color, select the **Color...** button to open the color selection dialog box.
6. Click on the color you want and then click **OK** to close the color selection dialog box.
7. Click **OK** to close the fill pattern editor.
8. Click the **Close** button to finish.

Add an Image to a Chart Title Frame

1. Specify which title you want to change in the drop-down list box.
2. Select the **Format** sub-tab.
3. Click the **Pattern...** button to open the fill pattern editor.
4. To add an image, select the **Browse...** button to open the file selection dialog box.
5. Choose the graphic image you want and then click **OK** to close the file selection dialog box.
6. Click **OK** to close the fill pattern editor.
7. Click the **Close** button to finish.

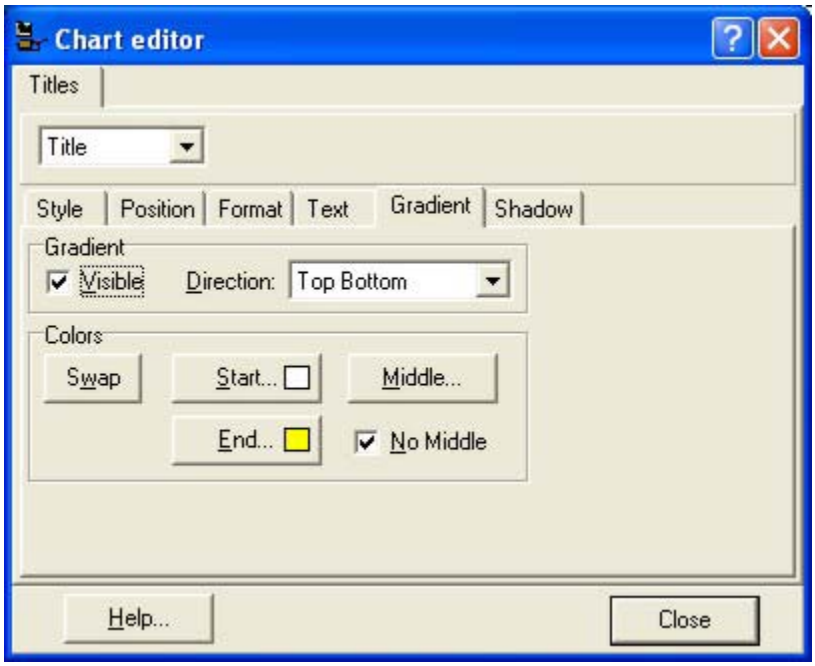
Change the Chart Title Font Properties



The chart title text editor tab

1. Specify which title you want to change in the drop-down list box.
2. Click the **Text** sub-tab to display the font properties.
3. Click the **Font…** button to display the Font Properties dialog box and select the **Font** (e.g. Arial, Times New Roman, etc), the **Font Style** (e.g. Italic, Bold, etc.), **Size**, and **Color** until the **Sample Text** has the appearance you want.
4. Click the **OK** button to close the Font Properties dialog box.
5. Click the **Color** button in the **Shadow…** group to display the Color dialog box and select from the **Basic Colors** to add a shadow to your text if desired.
6. Click the **OK** button to close the Color dialog box.
7. Click the spin edit boxes to adjust the horizontal and vertical aspects of the shadow or enter a value from the keyboard.
8. Click the **Inter-char spacing** spin edit box to adjust the character spacing or enter a value from the keyboard.
9. Click the **Close** button to finish.

Add a Gradient to a Chart Title Frame

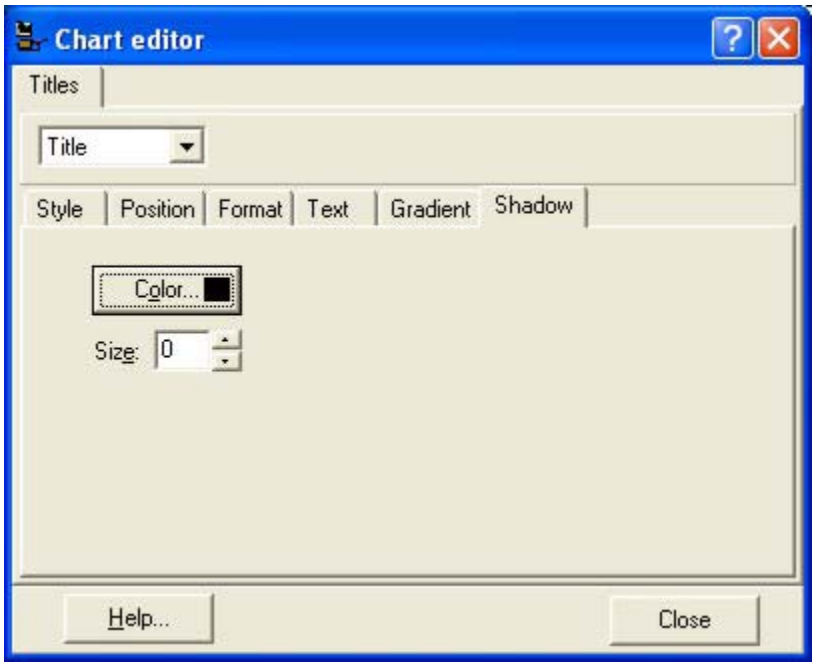


The chart title gradient editor tab

1. Specify which title you want to change in the drop-down list box.
2. Select the **Gradient** sub-tab.
3. Ensure the **Visible** box is checked.
4. Select a direction (eg. Left Right, Top Bottom) for the gradient from the **Direction** drop-down list.
5. In the **Colors** section of the dialog box, use the **Start**, **Middle** and **End** buttons to select the colors for the gradient. If no middle color is required, check the **No Middle** box.
6. To reverse the start and end colors, use the **Swap** button.
7. Click the **Close** button to finish.

Note : In order see the gradient you must make sure that the **Transparent** check box on the **Format** tab is unchecked.

Add an Shadow to a Chart Title Frame



The chart title shadow editor tab

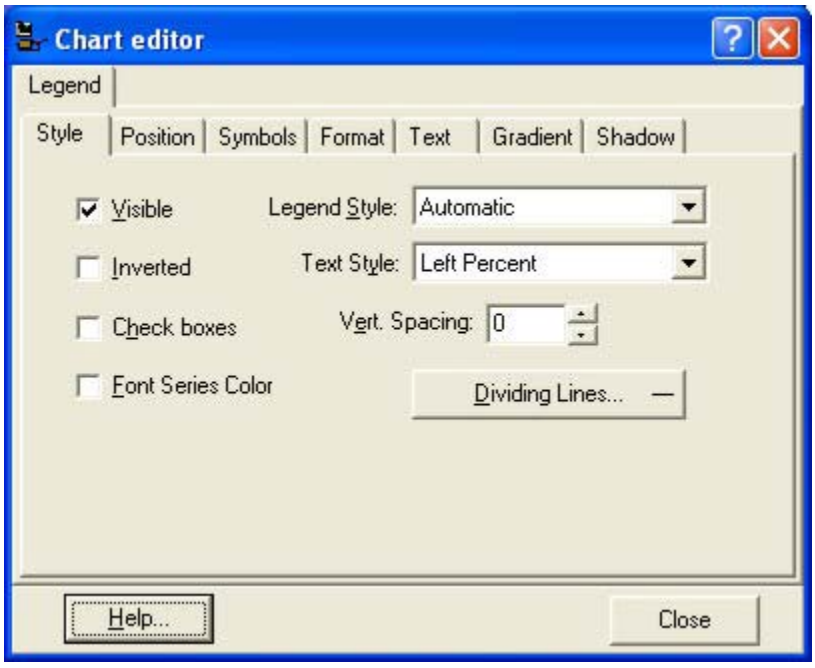
1. Specify which title you want to change in the drop-down list box.
2. Select the **Shadow** sub-tab.
3. Click the **Color** button to display the Color dialog box and select from the **Basic Colors** to add a shadow to your text if desired.
4. Click the **OK** button to close the Color dialog box.
5. Click the **Size** spin edit box to adjust the depth of the shadow or enter a value from the keyboard.
6. Click the **Close** button to finish.

Chart Legend Procedures

This section contains topics covering procedures that can be executed from the **Edit Legend** tab. The procedures outlined include a variety of chart legend formatting options such as outlines, three-dimensional appearance, position and fonts. The list below contains some general legend tips:

- To have no legend, un-check the box labeled **Visible**.
- To reverse the order in which series are placed in the legend, check the box labeled **Inverted**.
- To add check boxes beside each item in the legend, check the box labeled **Check boxes**.

- To change the color of the text in the legend to match the series in the graph, check the box labeled **Font Series Color**.



The chart legend style editor tab

Legend Styles

This section gives descriptions of the various legend styles that may be applied to chart legends in BioWin. Generally, the **Automatic** style is the **Legend Style** best suited for most series types.

There are two attributes that control the appearance of legends in BioWin. The **Legend Style** determines the possible *content* of the legend. The **Text Style** determines the *layout* of the legend content. The table below describes the possible legend styles.

Style	Description
Automatic	Generally, if there is only one series on your chart, this style is will show the series values. If there are two or more series, this style will show the names of the various series.
Series Names	Shows the names of all series in a chart. Since only names are shown in the legend, **Text Styles** do not apply.
Series Values	The legend shows information for only the first series in the chart.
Last Values	The legend shows information for the last point of each series in the chart.

Tip: To control the numeric format of your legend entries, change the series **Label Number Format**. To do this:

1. Right-click the chart and select **Edit Series**.
2. From the series drop list box, select the series corresponding to the legend entry you want to change the numeric formatting of.

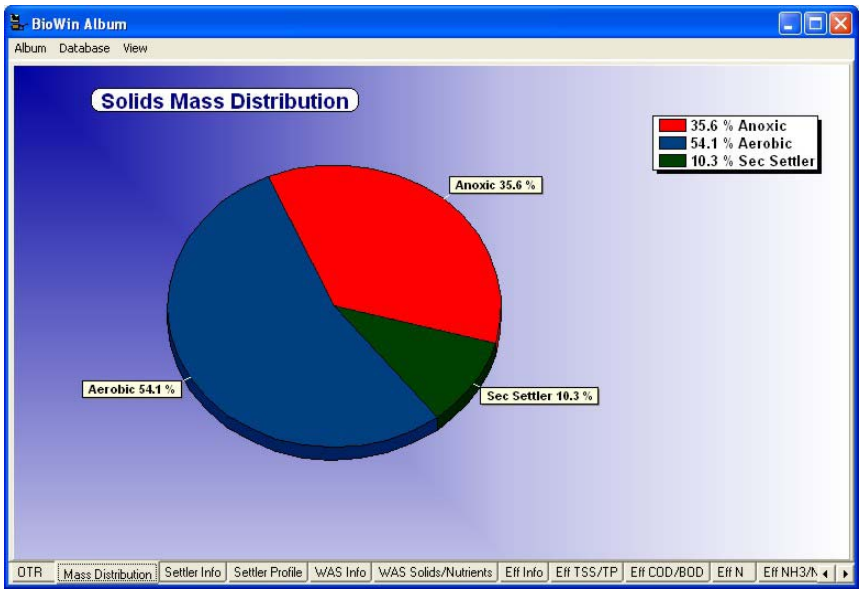
3. Click on the **Series** tab.
4. Click on the **General** sub-tab
5. In the **Formats** group, change the **Values** or **Percents** formatting to the style you want.
6. Click **OK** to finish.

Legend Text Styles

This section gives examples of the various legend text styles that may be applied to BioWin chart legends. The table below provides a simple example of the available legend text styles.

Style	Example
Plain	Unaerated
Right Value	Unaerated 37, 970.173
Left Value	37, 970.173 Unaerated
Right Percent	Unaerated 26.6 %
Left Percent	26.6 % Unaerated
X Value	0
Value	37, 970.173
Percent	26.6 %
X and Value	0 37, 970.173
X and Percent	0 26.6 %

For example, consider the process mass plot shown below. The legend shown in this picture has the **Left Percent** legend text style applied.



Process mass plot with Left Percent *Legend Text Style applied to legend*

Some of the legend text styles mentioned above refer to series "points". In BioWin charts, a point can refer to a bar of a bar series, or a pie slice of a pie series. Note that time series (e.g. Line and point series) points do not have names.

Tip: To control the numeric format of your legend entries, change the series **Label Number Format**. To do this:

1. Right-click the chart and select **Edit Series**.
2. From the series drop list box, select the series corresponding to the legend entry you want to change the numeric formatting of.
3. Click on the **Series** tab.
4. Click on the **General** sub-tab
5. In the **Formats** group, change the **Values** or **Percents** formatting to the style you want.
6. Click **OK** to finish.

Note: When you apply a **Legend Text Style** that involves percent, the individual point percentages are calculated as follows. The total sum of the point values is calculated. Then each individual point percentage is calculated by dividing the point value by the total sum of the points.

Change the Chart Legend Vertical Spacing

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Legend**.
2. Select the legend spacing you want from the **Vert Spacing** spin edit box.
3. Click the **Close** button to finish.

Add Dividing Lines to a Legend

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Legend**.
2. Click the **Dividing Lines...** button to open the border editor.
3. Make sure that the box labeled **Visible** is checked.
4. Select the dividing line style you want from the **Style** drop-down list
5. Select the dividing line width you want from the **Width** spin edit box.
6. To change the dividing line color, select the **Color...** button to open the color selection dialog box.
7. Click on the dividing line color you want and then click **OK** to close the color selection dialog box.
8. Click **OK** to close the border editor.
9. Click the **Close** button to finish.

Change the Legend Position

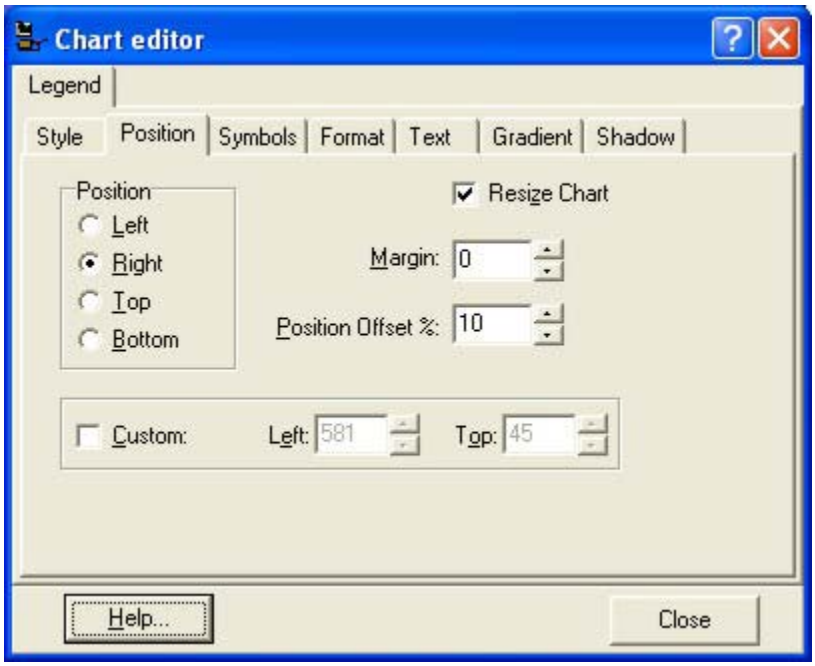


Chart legend position dialog box

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Legend**.
2. Click the **Position** sub-tab.
3. Make sure that the box labeled **Resize Chart** is checked, so that the chart will reposition itself according to the legend position you select.
4. In the **Position** group, use the radio buttons to choose **Left**, **Right**, **Top** or **Bottom**.
5. Adjust the amount of space between the legend and the chart using the **Margin** spin edit box.
6. Fine-tune the position of the legend using the **Position Offset %** spin edit box.
7. Set up a custom location by checking the box labeled **Custom** and adjusting the **Left** and **Top** spin edit boxes.
8. Click the **Close** button to finish.

Change the Appearance of Legend Symbols

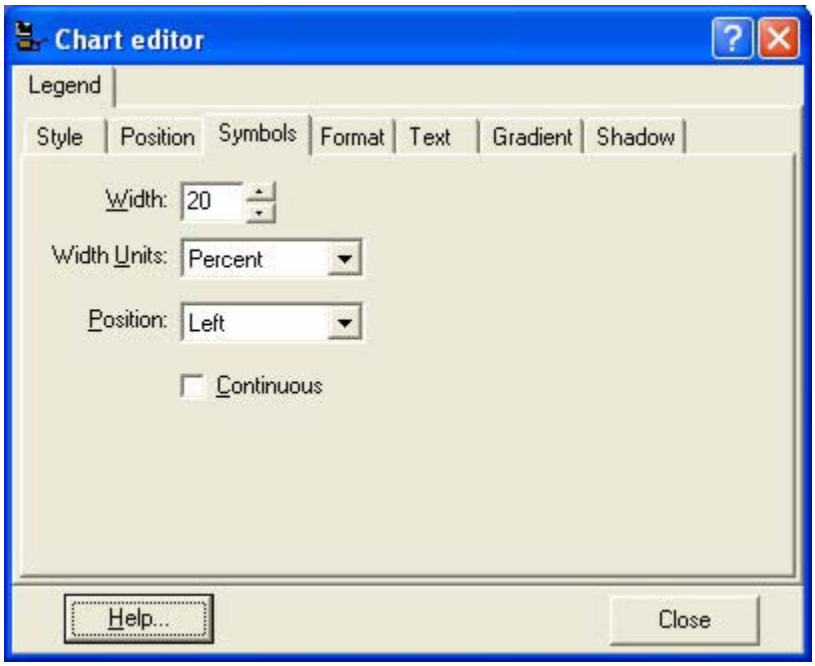


Chart legend symbols dialog box

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Legend**.
2. Click the **Symbols** sub-tab.
3. Adjust the size of the legend symbols using the **Width** spin edit box.
4. Choose between **Percent** and **Pixels** in the **Width Units** drop-down list.
5. Position the symbols on the **Left** or **Right** of the legend text with the **Position** drop-down list.
6. Remove the space between symbols by checking the box labeled **Continuous**. Note this option is only effective if the legend is located on the left or right of the chart.
7. Click the **Close** button to finish.

Change the Chart Legend Color and Pattern

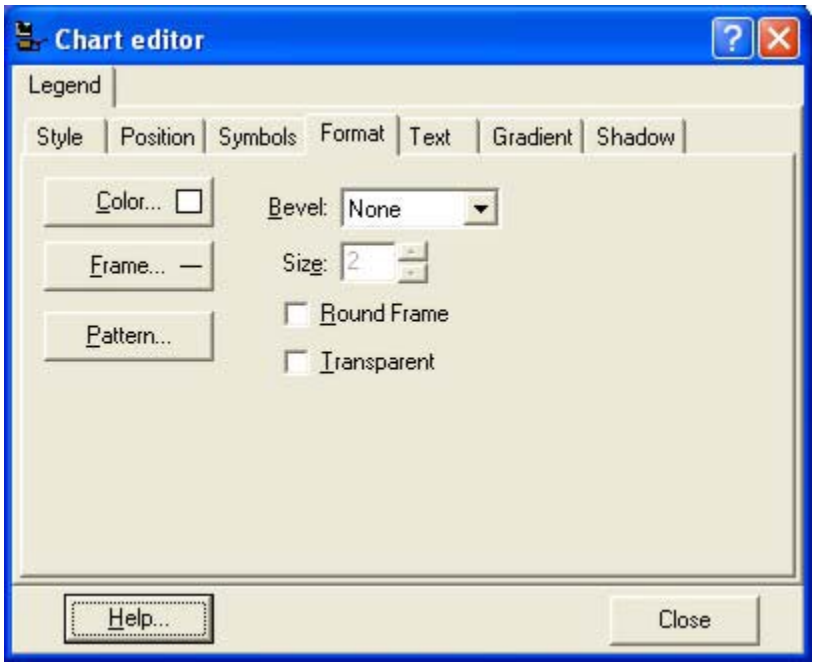


Chart legend format dialog box

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Legend**.
2. Click the **Format** sub-tab.
3. Select the **Color...** button to open the color selection dialog box.
4. Click on the legend background color you want and then click **OK** to close the color selection dialog box.
5. If a background pattern is required, select the **Pattern…** button to open the pattern color editor dialog box.
6. Select the fill style you want from the list box.
7. To change the fill color, select the **Color...** button to open the color selection dialog box.
8. Click on the color you want and then click **OK** to close the color selection dialog box.
9. Click **OK** to close the pattern color editor.
10. Click the **Close** button to finish.

Add a Frame to the Chart Legend

Border editor dialog box

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Legend**.
2. Click the **Format** sub-tab.
3. Select the **Frame...** button to open the border editor dialog box.
4. Make sure that the box labeled **Visible** is checked.
5. Select the frame style you want from the **Style** drop-down list.
6. Select the frame width you want from the **Width** spin edit box.
7. To change the frame color, select the **Color...** button to open the color selection dialog box.
8. Click on the frame color you want and then click **OK** to close the color selection dialog box.
9. Click **OK** to close the border editor.
10. If a rounded box is required, check the box labeled **Round Frame**.
11. If a three-dimensional effect is required, select **Raised** or **Lowered** from the **Bevel** drop-down list. The depth of this effect can be adjusted with the **Size** spin edit box.
12. Click the **Close** button to finish.

Change the Chart Legend Font

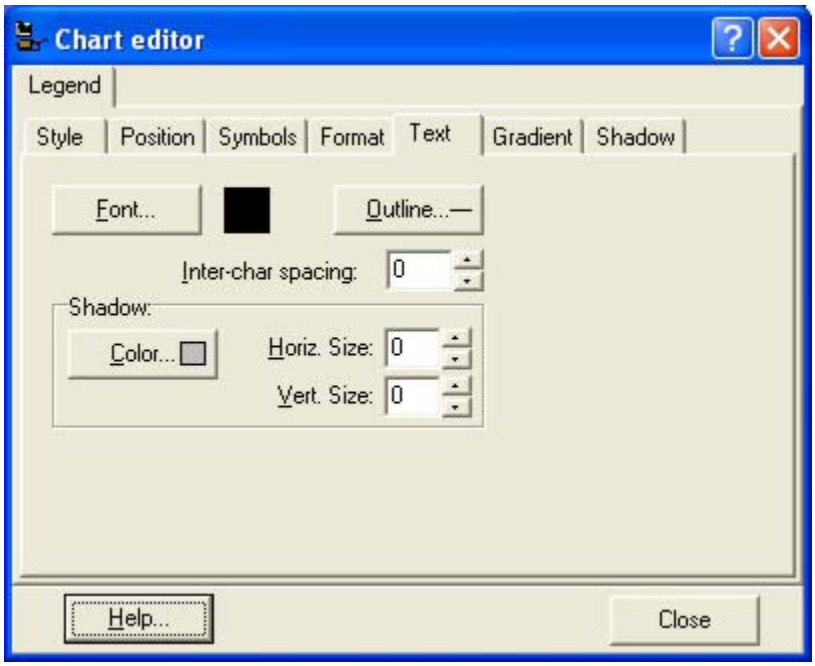


Chart legend text dialog box

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Legend**.
2. Click the **Text** sub-tab.
3. Click the **Font...** button to open the font properties dialog box.
4. Specify the **Font** (e.g. Arial, Times New Roman, etc), the **Font Style** (e.g. Italic, Bold, etc.), **Size**, and **Color** until the **Sample Text** has the appearance you want.
5. Click the **OK** button to close the font properties dialog box.
6. Click the **Color** button in the **Shadow…** group to display the **Color** dialog box and select from the **Basic Colors** to add a shadow to your text if desired.
7. Click the **OK** button to close the Color dialog box.
8. Click the spin edit boxes to adjust the horizontal and vertical aspects of the shadow or enter a value from the keyboard.
9. Click the **Inter-char spacing** spin edit box to adjust the character spacing or enter a value from the keyboard.
10. Click the **Close** button to finish.

Add a Color Gradient to the Chart Legend

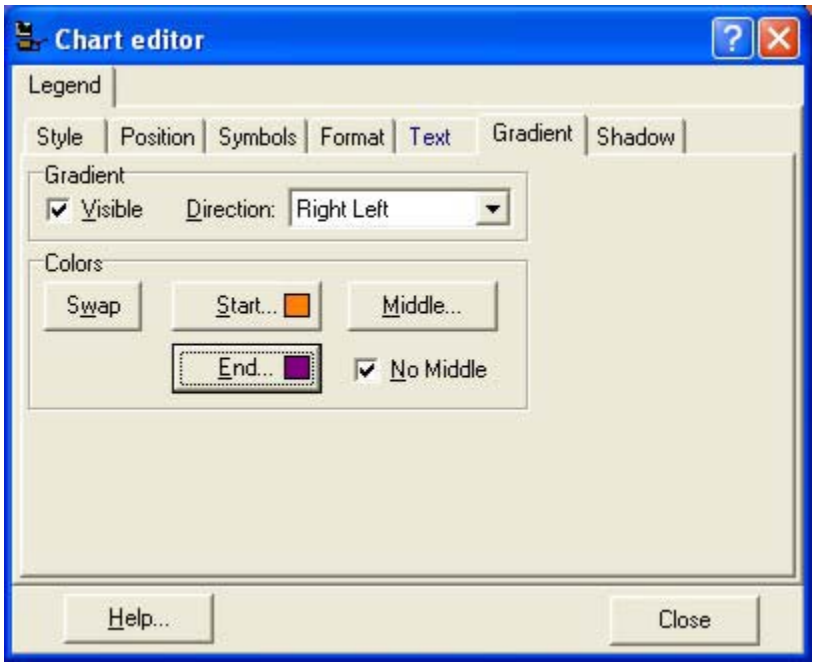


Chart legend gradient dialog box

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Legend**.
2. Select the **Gradient** sub-tab.
3. Ensure the **Visible** box is checked.
4. Select a direction (eg. Left Right, Top Bottom) for the gradient from the **Direction** drop-down list.
5. In the **Colors** section of the dialog box, use the **Start**, **Middle** and **End** buttons to select the colors for the gradient. If no middle color is required, check the **No Middle** box.
6. To reverse the start and end colors, use the **Swap** button.
7. Click the **Close** button to finish.

Add a Shadow to the Chart Legend

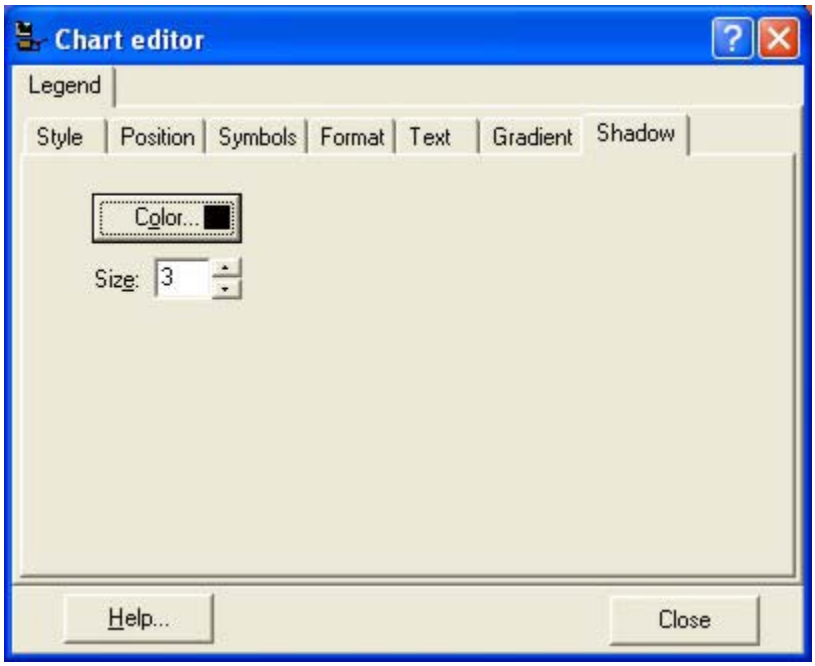


Chart legend shadow dialog box

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Legend**.
2. Select the **Shadow** sub-tab.
3. Click the **Color** button to display the Color dialog box and select from the **Basic Colors** to add a shadow to your text if desired.
4. Click the **OK** button to close the Color dialog box.
5. Click the **Size** spin edit box to adjust the depth of the shadow or enter a value from the keyboard.
6. Click the **Close** button to finish.

Other Chart Options and Procedures

This section consists of procedures and options that change the general appearance of charts. All of these procedures can be executed after right clicking on the chart to open the **Edit** menu, then clicking **Edit Options**.

Chart Options

This section consists of procedures that can be executed from the **Chart** tab of the **Edit Options** menu choice.

Change the Series Order

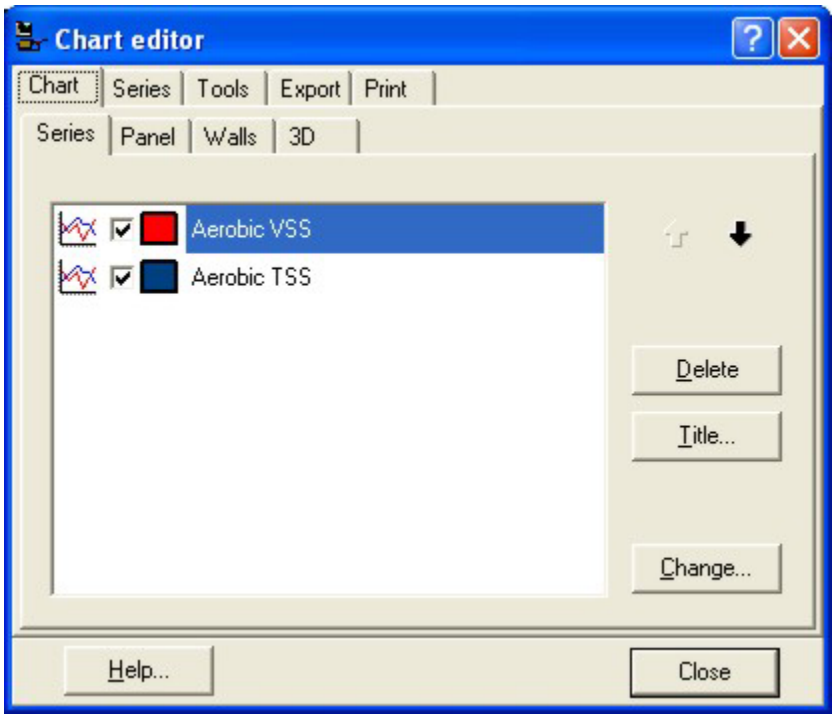


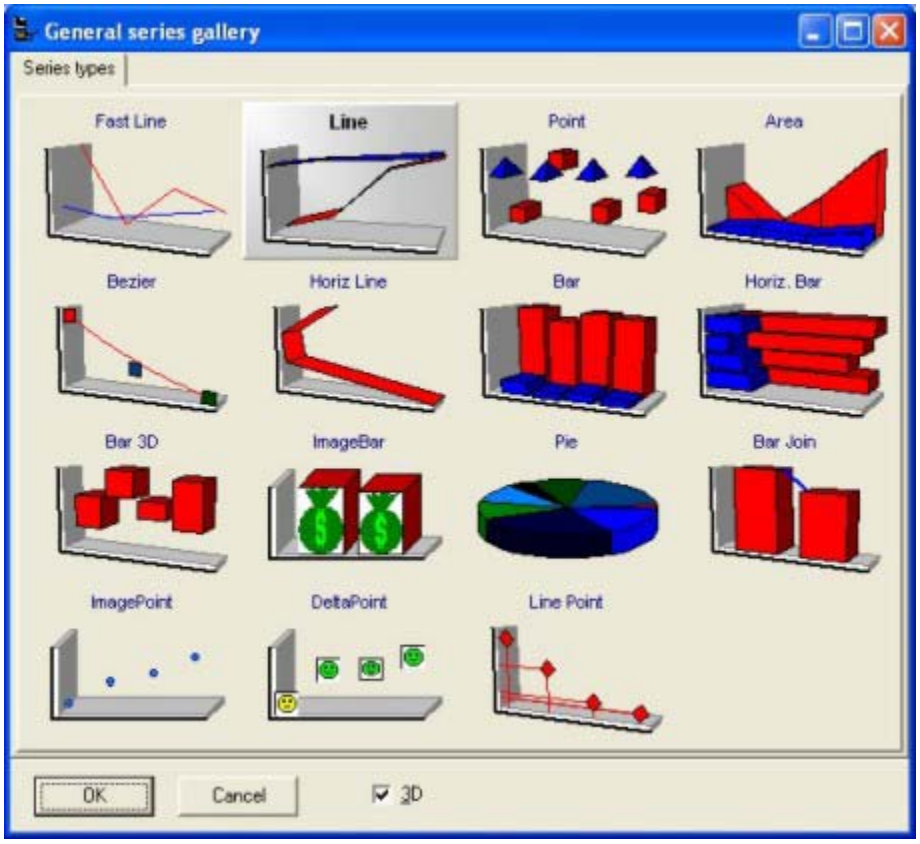
Chart series options dialog box

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Series** tab.
3. Click on the series that needs to be moved in the series list.
4. Click the  or  button to change the order of the selected item in the list.
5. Click the **Close** button to finish.

Note: Use the check boxes beside the series to temporarily remove them from the chart display. This dialog box can also be used to **Delete** the selected series and edit the **Title…** text.

Change the Series Type

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Series** tab.
3. Click on the series that needs to be changed in the series list.
4. Click the **Change…** button to display the general series gallery dialog box.



General series type dialog box

5. Uncheck the box labeled **3D** to display two dimensional chart types if required.
6. Click any chart type that you would like to use for the series.
7. Click **OK** to close the series type dialog box.
8. Click the **Close** button to finish.

Chart Panel Options

This section consists of procedures that can be executed from the **Panel** sub-tab of the **Chart** tab in the **Edit Options** menu choice. It includes procedures for changing the background appearance and border options for your chart.

Changing the Chart Background Color

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Chart** tab.
3. Click the **Panel** sub-tab to access the panel options.
4. Click the **Background** sub-tab.

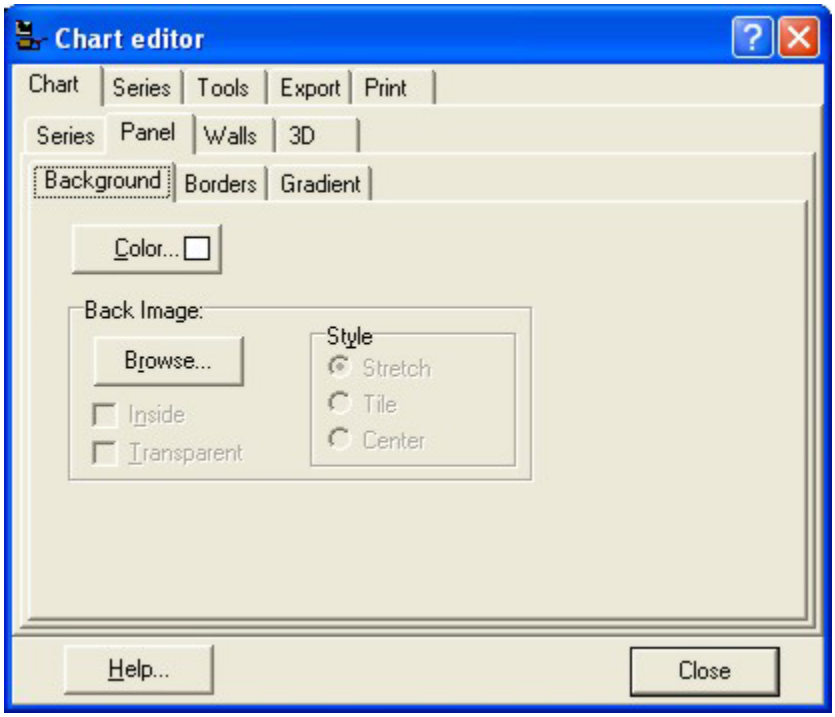


Chart panel background dialog box

5. Click the **Color…** button to display the Color dialog box and select from the **Basic Colors** to add a color to the chart background.
6. Click the **OK** button to close the Color dialog box.
7. Click the **Close** button to finish.

Note: This option can also be used to add an image to the chart background using the **Browse**… button in the section labeled **Back Image**. This opens a file selection dialog box from which you can choose a graphic image (such as a company logo) to place behind the chart. Once an image is chosen, options from the Style radio button list are available to control the display of the image. Choose between **Stretch**, **Tile** and **Center**.

Adding a Border to the Chart

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Chart** tab.
3. Click the **Panel** sub-tab to access the panel options.
4. Click the **Borders** sub-tab.

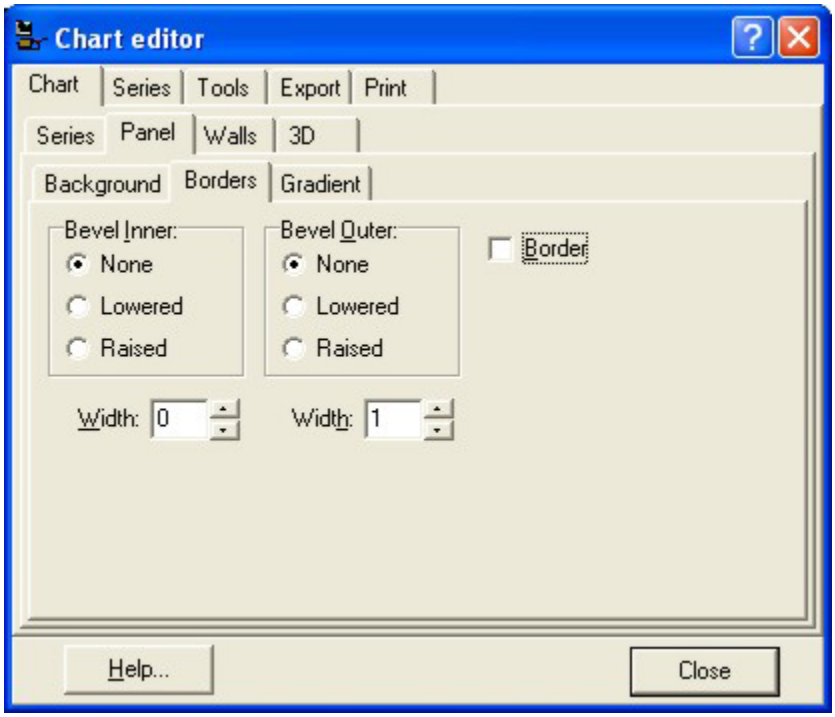
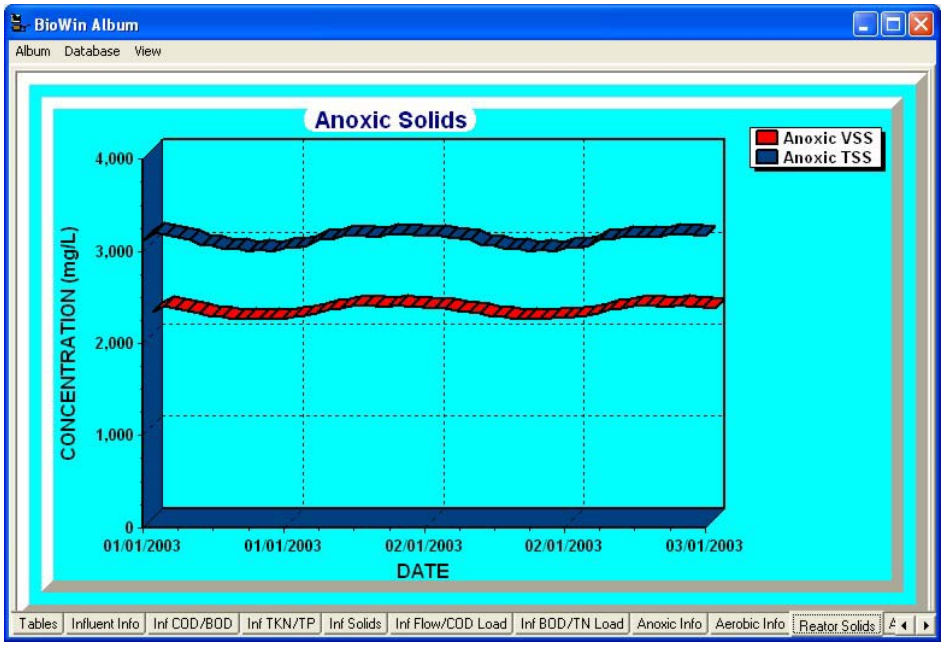


Chart panel borders dialog box

5. Click the check box labeled **Border** to add a single line around the entire chart.
6. Use the radio buttons in the **Bevel Inner** and **Bevel Outer** sections to add or remove **Lowered** or **Raised** bevel effects.
7. Adjust the width of the bevel effects using the **Width** spin edit boxes.
8. Click the **Close** button to finish.

Below is an example of a chart with the effects chosen in the dialog box above:



An example of a chart with beveled inner and outer borders

Adding a Gradient to the Chart Background

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Chart** tab.
3. Click the **Panel** sub-tab to access the panel options.
4. Click the **Gradient** sub-tab.

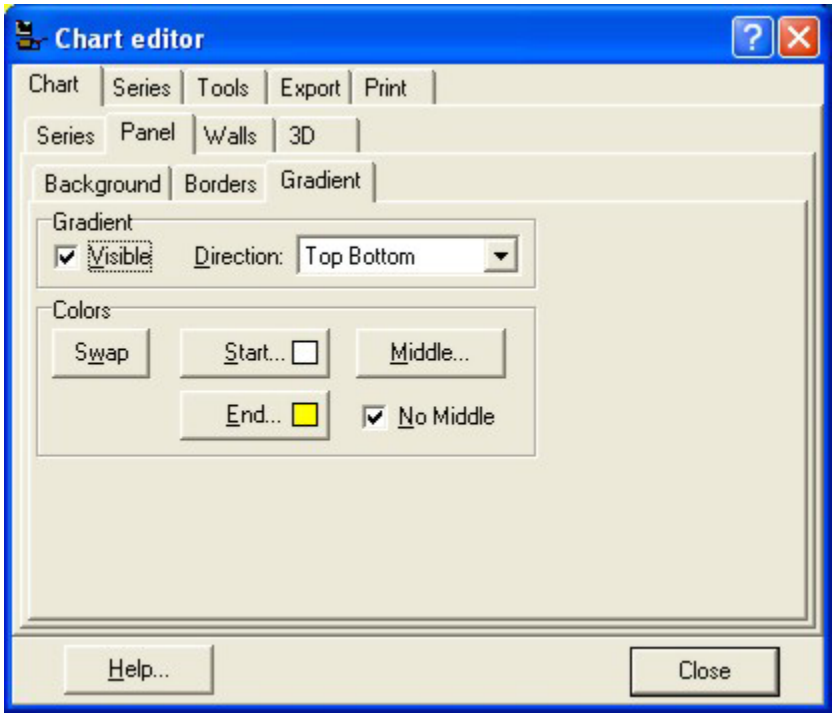


Chart panel gradient dialog box

5. Ensure the **Visible** box is checked.
6. Select a direction (eg. Left Right, Top Bottom) for the gradient from the **Direction** drop-down list.
7. In the **Colors** section of the dialog box, use the **Start**, **Middle** and **End** buttons to select the colors for the gradient. If no middle color is required, check the **No Middle** box.
8. To reverse the start and end colors, use the **Swap** button.
9. Click the **Close** button to finish.

Adjust the Appearance of Chart Walls

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Chart** tab.
3. Click the **Walls** sub-tab to access the wall display options.

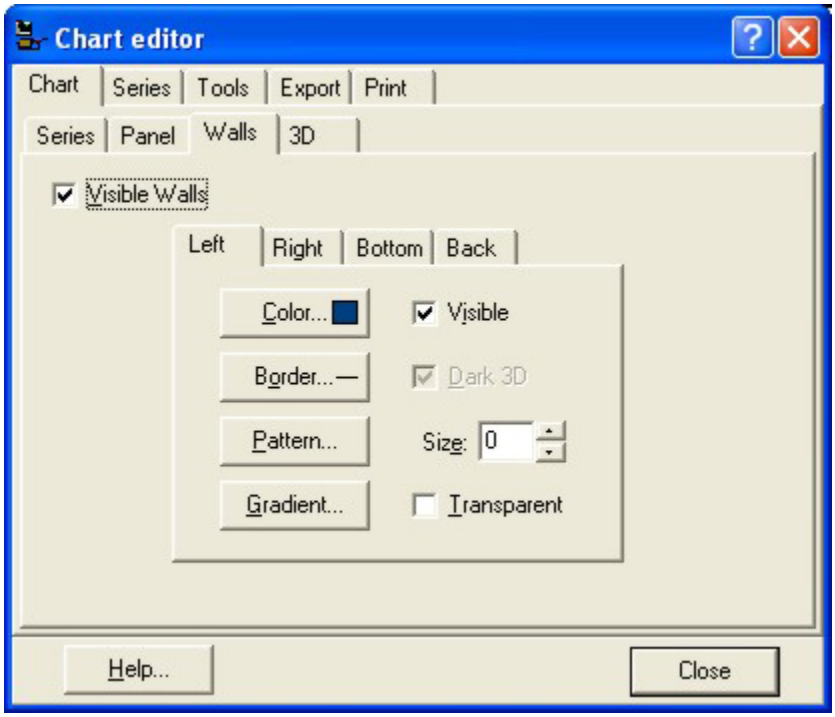


Chart walls dialog box

4. If you want the walls of the chart to be transparent, un-check the box labeled **Visible Walls**. If you want to see them, make sure that this box is checked.
5. To change the color of a three-dimensional wall, select the desired wall by clicking on the appropriate sub-tab **(Left, Right, Bottom or Back)**, and select the **Color...** button to open the color selection dialog box.
6. Click on the wall color you want and then click **OK** to close the color selection dialog box.

7. To change the border of a three-dimensional wall, select the desired wall by clicking on the appropriate sub-tab **(Left, Right, Bottom or Back)**, and select the **Border...** button to open the border color editor.
8. Make sure that the box labeled **Visible** is checked.
9. Select the wall border style you want from the **Style** radio button group.
10. Select the wall border width you want from the **Width** spin edit box.
11. To change the wall border color, select the **Color...** button to open the color selection dialog box.
12. Click on the wall border color you want and then click **OK** to close the color selection dialog box.
13. Click **OK** to close the border color editor.
14. To change the pattern of a three-dimensional wall, select the desired wall by clicking on the appropriate sub-tab **(Left, Right, Bottom or Back)**, and select the **Pattern...** button to open the pattern selection dialog box.
15. Click on the wall pattern you want and then click **OK** to close the pattern selection dialog box.
16. To change the gradient of a three-dimensional wall, select the desired wall by clicking on the appropriate sub-tab **(Left, Right, Bottom or Back)**, and select the **Gradient...** button to open the gradient selection dialog box.
17. Ensure the **Visible** box is checked.
18. Select a direction (eg. Left Right, Top Bottom) for the gradient from the **Direction** drop-down list.
19. In the **Colors** section of the dialog box, use the **Start**, **Middle** and **End** buttons to select the colors for the gradient. If no middle color is required, check the **No Middle** box.
20. To reverse the start and end colors, use the **Swap** button.
21. Click **OK** to close the gradient selection dialog box.
22. To change the size of a three-dimensional wall, select the desired wall by clicking on the appropriate sub-tab **(Left, Right, Bottom or Back)**, and adjust the value in the **Size** spin edit box.
23. To remove any of these options from an individual wall, click the appropriate sub-tab **(Left, Right, Bottom or Back)**, and remove the check from the box labeled **Visible**.
24. Click the **Close** button to finish.

Adjust Chart Three-Dimensional Appearance

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Chart** tab.
3. Click the **3D** sub-tab to access the 3D display options.

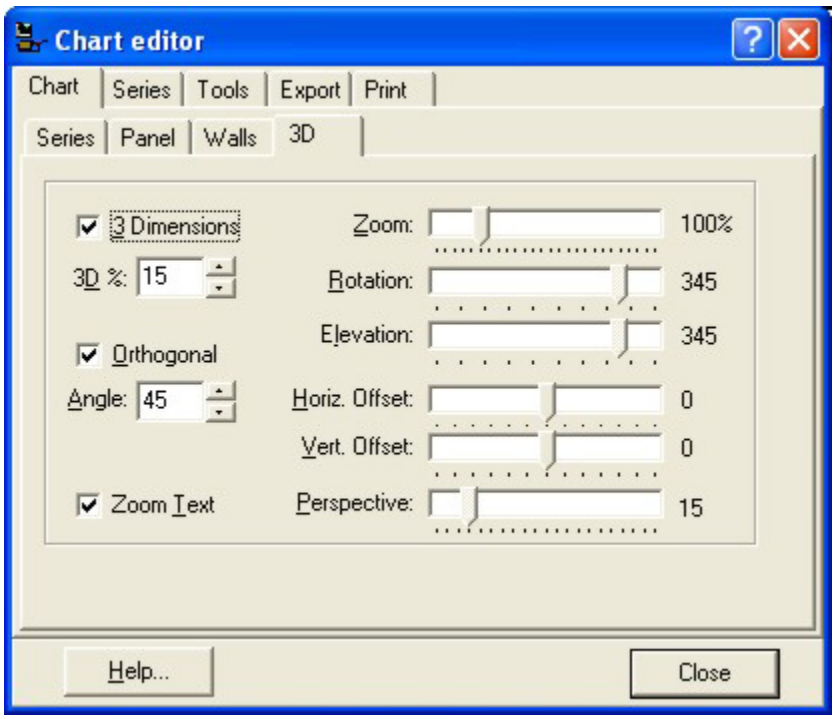


Chart 3D dialog box

4. If you want your chart to have a three-dimensional appearance, ensure that the box labeled **3 Dimensions** is checked.
5. If you do **not** want your chart to have a three-dimensional appearance, ensure that the box labeled **3 Dimensions** is cleared.
6. To adjust the three-dimensional depth of the chart, increase or decrease the value in the **3D %** spin edit box.
7. Checking the **Orthogonal** check box and increasing or decreasing the value in the **Angle** spin edit box can further adjust the three-dimensional appearance.
8. To adjust the size of the chart on the page, drag the point in the section labeled **Zoom**.
9. To rotate the chart on the page, first uncheck the box labeled **Orthogonal**, and then drag the point in the section labeled **Rotation**.
10. To change the elevation of the chart on the page, first uncheck the box labeled **Orthogonal**, and then drag the point in the section labeled **Elevation**.
11. The chart can be moved to the left or right on the page by dragging the point in the section labeled **Horiz Offset**.
12. The chart can be moved up or down on the page by dragging the point in the section labeled **Vert Offset**.
13. Click the **Close** button to finish.

Series Options

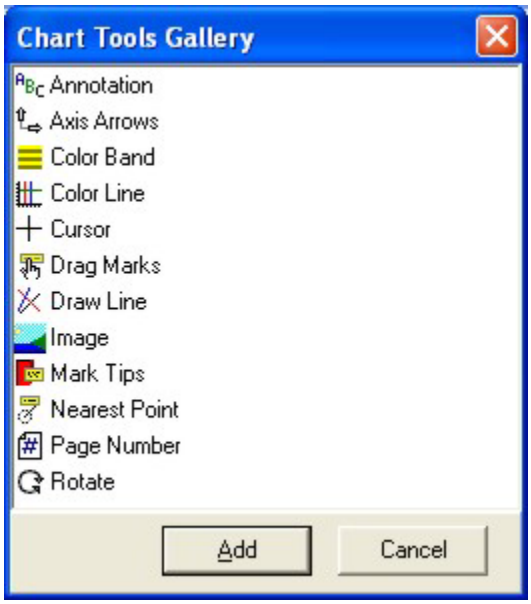
For information on the procedures that can be executed from the **Series** tab of the **Edit Options** menu choice, please see "*Series Formatting Procedures*".

Tools Options

This section consists of procedures that can be executed from the **Tools** tab of the **Edit Options** menu choice. It includes procedures for customizing your graph including hand-drawn lines, annotations and page numbering. This section is organized so that its sub-topics appear in the same order as the options that appear when all tools are added to the **Tools** dialog box.

Adding a Tool

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list



Tools gallery dialog box

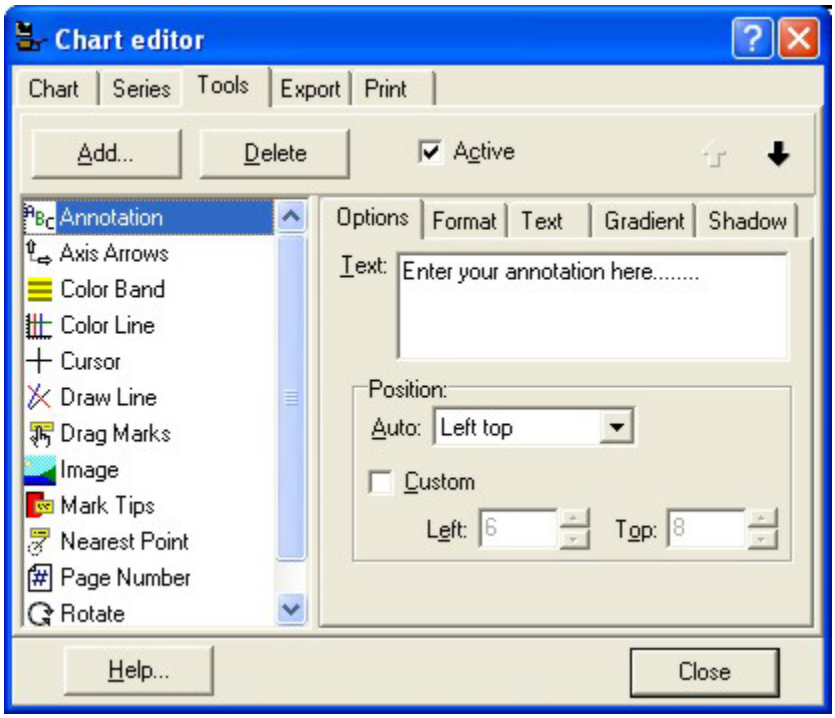
4. Click on the tool you would like to use (eg. **Annotation**).
5. Click the **Add** button to finish.

Repeat these steps as necessary to add the tools you would like to use to the **Tools** dialog box.

Adding an Annotation to the Chart

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.

3. Click the **Add…** button to access the tools list and **Add** the **Annotation** tool if necessary. See the **Adding a Tool** section for more information.

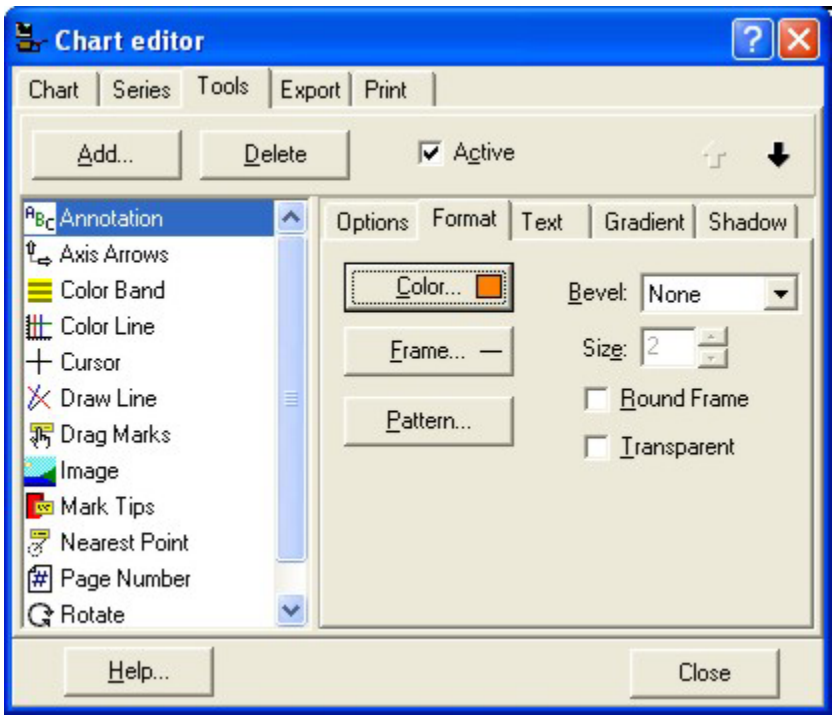


Tools annotation options dialog box

4. In the section labeled **Text**, type the annotation you require, pressing the Enter key each time you want to start a new line. The annotation box in the graph will automatically adjust to the width and height of the lines you type.
5. To change the position of the annotation within the graph, use the **Auto** drop-down list in the **Position** section to choose a location (**Left top**, **Left bottom**, **Right top**, **Right bottom**).
6. To set a custom location for the annotation, check the box labeled **Custom**, and adjust the position using the **Left** and **Top** spin edit boxes.
7. Click the **Close** button to finish.

Changing the Annotation Format

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Annotation** tool if necessary. See the **Adding a Tool** section for more information.
4. Click the **Format** sub-tab.



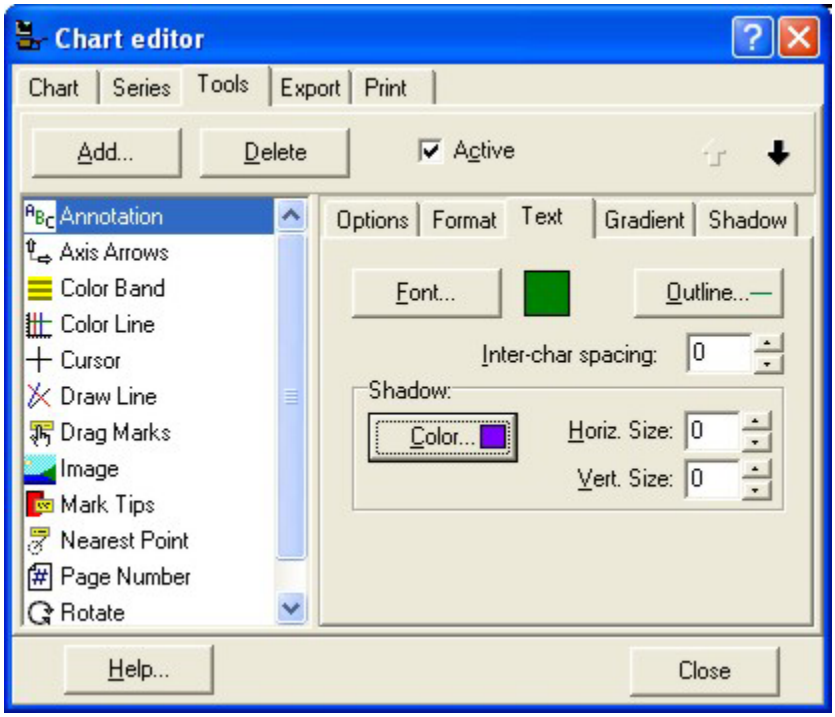
Tools annotation format dialog box

5. Select the **Color...** button to open the color selection dialog box.
6. Click on the background color you want and then click **OK** to close the color selection dialog box.
7. Click **OK** to close the border color editor.
8. To adjust the appearance of the annotation frame, select the **Frame...** button to open the color selection dialog box.
9. Make sure that the box labeled **Visible** is checked.
10. Select the frame style you want from the **Style** drop-down list.
11. Select the frame width you want from the **Width** spin edit box.
12. To change the frame color, select the **Color...** button to open the color selection dialog box.
13. Click on the frame color you want and then click **OK** to close the color selection dialog box.
14. Click **OK** to close the border editor.
15. If a rounded box is required, check the box labeled **Round Frame**.
16. If a three-dimensional effect is required, select **Raised** or **Lowered** from the **Bevel** drop-down list. The depth of this effect can be adjusted with the **Size** spin edit box.
17. If a background pattern is required, select the **Pattern…** button to open the pattern color editor dialog box.
18. Select the fill style you want from the list box.

19. To change the fill color, select the **Color...** button to open the color selection dialog box.
20. Click on the color you want and then click **OK** to close the color selection dialog box.
21. Click **OK** to close the pattern color editor.
22. Click the **Close** button to finish.

Changing the Annotation Text

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Annotation** tool if necessary. See the **Adding a Tool** section for more information.
4. Click the **Text** sub-tab.



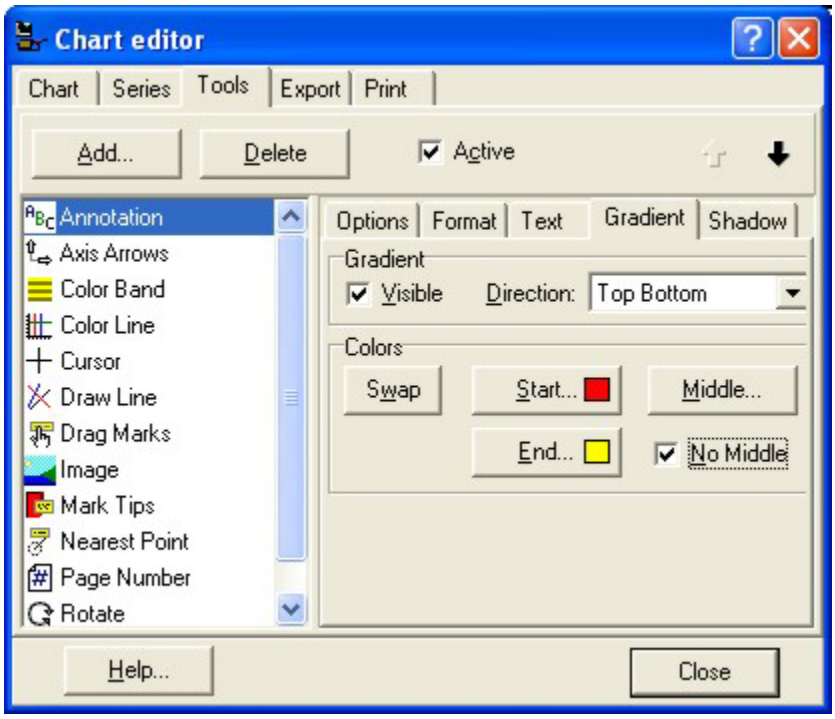
Tools annotation text dialog box

5. Click the **Font…** button to display the Font Properties dialog box and select the **Font** (e.g. Arial, Times New Roman, etc), the **Font Style** (e.g. Italic, Bold, etc.), **Size**, and **Color** until the **Sample Text** has the appearance you want.
6. Click the **OK** button to close the Font Properties dialog box.
7. Click the **Color** button in the **Shadow…** group to display the **Color** dialog box and select from the **Basic Colors** to add a shadow to your text if desired.
8. Click the **OK** button to close the **Color** dialog box.

9. Click the spin edit boxes to adjust the horizontal and vertical aspects of the shadow or enter a value from the keyboard.
10. Click the **Inter-char spacing** spin edit box to adjust the character spacing or enter a value from the keyboard.
11. Click the **Close** button to finish.

Changing the Annotation Gradient

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Annotation** tool if necessary. See the **Adding a Tool** section for more information.
4. Click the **Gradient** sub-tab.

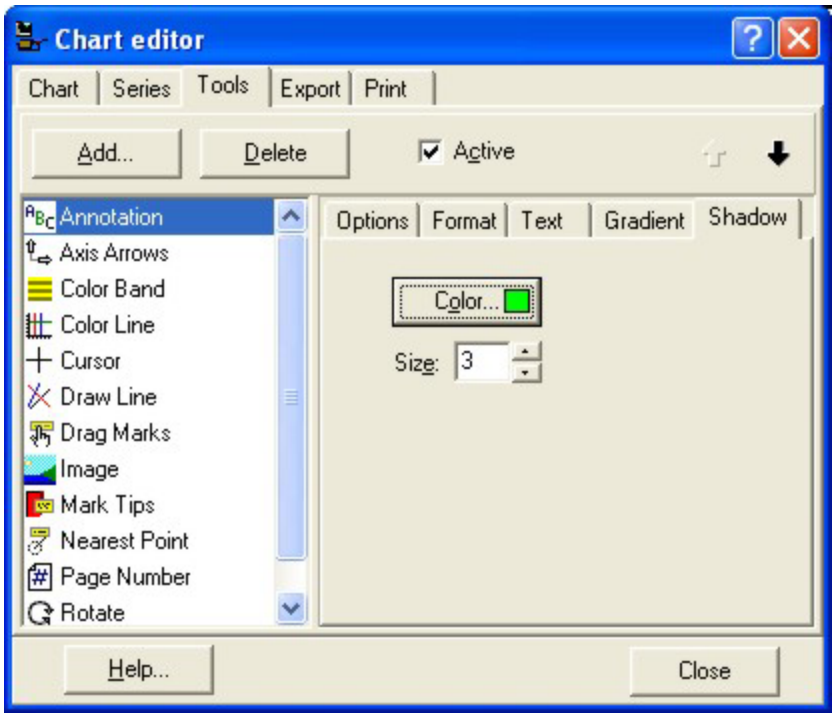


Tools annotation gradient dialog box

5. Ensure the **Visible** box is checked.
6. Select a direction (eg. Left Right, Top Bottom) for the gradient from the **Direction** drop-down list.
7. In the **Colors** section of the dialog box, use the **Start**, **Middle** and **End** buttons to select the colors for the gradient. If no middle color is required, check the **No Middle** box.
8. To reverse the start and end colors, use the **Swap** button.
9. Click the **Close** button to finish.

Changing the Annotation Shadow

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Annotation** tool if necessary. See the **Adding a Tool** section for more information.
4. Click the **Shadow** sub-tab.

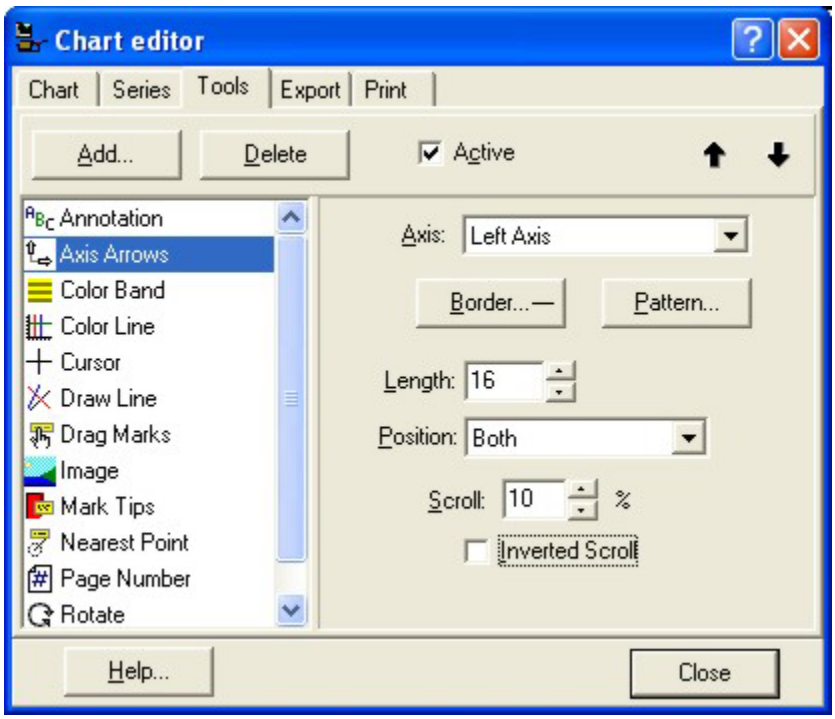


Tools annotation shadow dialog box

5. Click the **Color** button to display the Color dialog box and select from the **Basic Colors** to add a shadow to your text if desired.
6. Click the **OK** button to close the Color dialog box.
7. Click the **Size** spin edit box to adjust the depth of the shadow or enter a value from the keyboard.
8. Click the **Close** button to finish.

Adding Axis Arrows to the Chart

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Axis Arrows** tool if necessary. See the **Adding a Tool** section for more information.



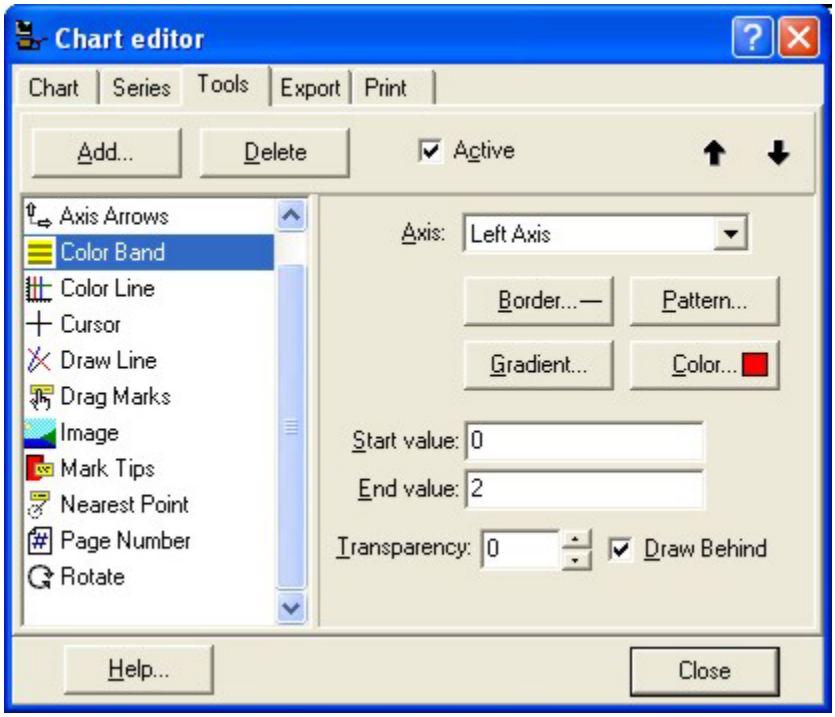
Tools axis arrows dialog box

4. To choose the location of the axis arrows, select from the **Axis** drop-down list (**Left axis**, **Right axis**, **Top axis** or **Bottom axis**).
5. To change the color of the axis arrows, click the **Border**… button to open the border editor dialog box.
6. Make sure that the box labeled **Visible** is checked.
7. Select the arrow width you want from the **Width** spin edit box.
8. To change the arrow color, select the **Color...** button to open the color selection dialog box.
9. Click on the arrow color you want and then click **OK** to close the color selection dialog box.
10. Click **OK** to close the border editor dialog box.
11. To change the axis arrow pattern, click the **Pattern…** button to open the pattern selection dialog box.
12. Click to select the pattern you require from the pattern list.
13. To change the pattern color, select the **Color...** button to open the color selection dialog box.
14. Click on the pattern color you want and then click **OK** to close the color selection dialog box.
15. Click OK to close the pattern selection dialog box.
16. To adjust the length of the axis arrows, increase or decrease the value in the **Length** spin edit box.
17. To choose the location for the axis arrows, select from the **Location** drop-down list (**Start**, **End** or **Both**).

18. To control the amount by which the axis is scrolled when the arrows are clicked on, change the **Scroll** percentage setting. You can also change the scrolling direction by checking the **Inverted Scroll** box.
19. Click the **Close** button to finish.

Adding a Color Band to the Chart

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Color Band** tool if necessary. See the **Adding a Tool** section for more information.



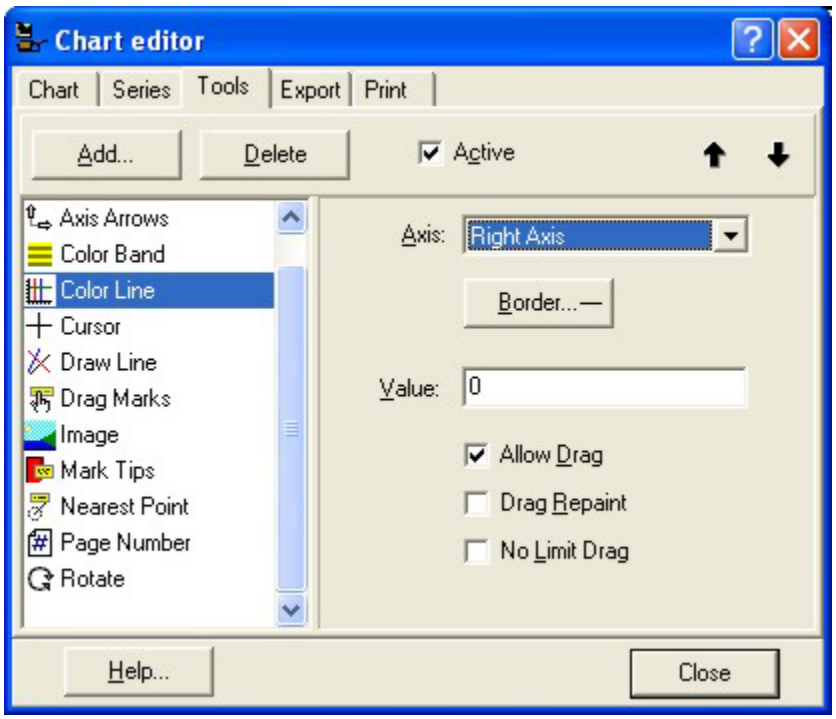
Tools color band dialog box

4. To choose the location for the color band, select from the **Axis** drop-down list (**Left axis**, **Right axis**, **Top axis** or **Bottom axis**).
5. To change the color of the color band, click the **Border**… button to open the border editor dialog box.
6. Make sure that the box labeled **Visible** is checked.
7. Select the border width you want from the **Width** spin edit box.
8. To change the border color, select the **Color...** button to open the color selection dialog box.
9. Click on the border color you want and then click **OK** to close the color selection dialog box.
10. To change the border style, select from the **Style** drop-down list.

11. Click **OK** to close the border editor dialog box.
12. To change the color band pattern, click the **Pattern…** button to open the pattern selection dialog box.
13. Click to select the pattern you require from the pattern list.
14. To change the pattern color, select the **Color...** button to open the color selection dialog box.
15. Click on the pattern color you want and then click **OK** to close the color selection dialog box.
16. Click OK to close the pattern selection dialog box.
17. To change the gradient of a color band, select the **Gradient...** button to open the gradient selection dialog box.
18. Ensure the **Visible** box is checked.
19. Select a direction (eg. Left Right, Top Bottom) for the gradient from the **Direction** drop-down list.
20. In the **Colors** section of the dialog box, use the **Start**, **Middle** and **End** buttons to select the colors for the gradient. If no middle color is required, check the **No Middle** box.
21. To reverse the start and end colors, use the **Swap** button.
22. Click **OK** to close the gradient selection dialog box.
23. To change the color of a color band, select the **Color...** button to open the color selection dialog box.
24. Click on the color you want and then click **OK** to close the color selection dialog box.
25. To adjust the position of the color band on the selected axis, increase or decrease the values in the **Start value** and **End value** boxes.
26. To draw the color band behind the graph, ensure the **Draw Behind** box is checked.
27. If **Draw Behind** is unchecked, adjust the transparency of the color band using the **Transparency** spin edit box.
28. Click the **Close** button to finish.

Adding a Color Line to the Chart

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Color Line** tool if necessary. See the **Adding a Tool** section for more information.



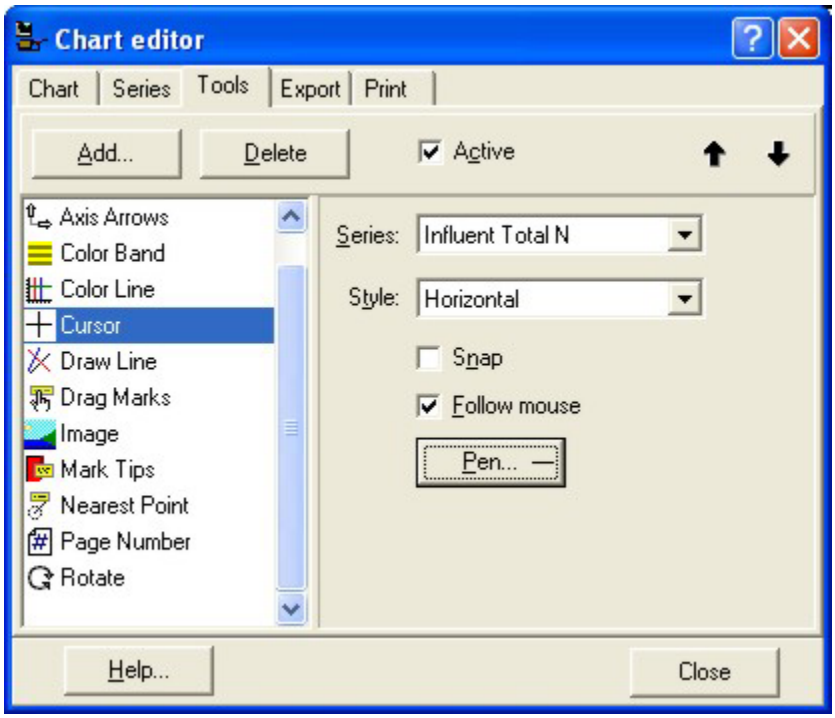
Tools color line dialog box

4. To choose the axis location for the color line, select from the **Axis** drop-down list (**Left axis**, **Right axis**, **Top axis** or **Bottom axis**).
5. To change the color of the color line, click the **Border**… button to open the border editor dialog box.
6. Make sure that the box labeled **Visible** is checked.
7. Select the border width you want from the **Width** spin edit box.
8. To change the border color, select the **Color...** button to open the color selection dialog box.
9. Click on the border color you want and then click **OK** to close the color selection dialog box.
10. To change the border style, select from the **Style** drop-down list.
11. Click **OK** to close the border editor dialog box.
12. To adjust the position of the color line on the selected axis, increase or decrease the value in the **Value** box.
13. To reposition the color line inside the graph using the mouse, check the box labeled **Allow Drag**. The **No Limit Drag** check box will allow you to position the color line anywhere inside or outside the graph.

Adding a Cursor to the Chart

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.

3. Click the **Add…** button to access the tools list and **Add** the **Cursor** tool if necessary. See the **Adding a Tool** section for more information.



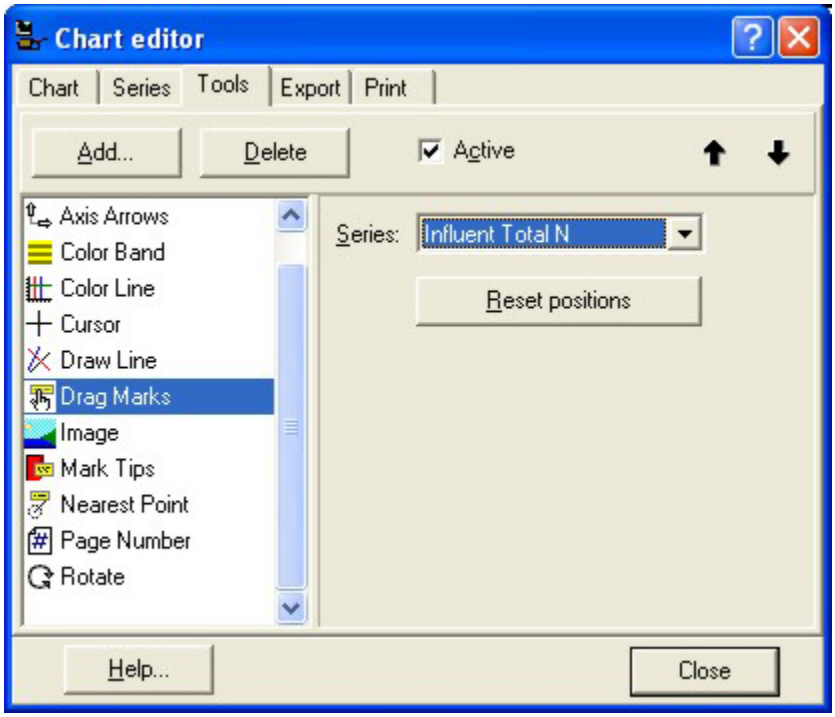
Tools cursor dialog box

4. To change the style of the cursor, select between **Horizontal**, **Vertical** and **Both** from the **Style** drop-down list.
5. The location of the cursor can be set to follow your mouse if you check the box labeled **Follow Mouse**. If this box is unchecked, you must move the cursor lines manually by dragging.
6. The cursor will snap to the data points on the selected axis if you check the box labeled **Snap**.
7. To change the appearance of the cursor, click the **Pen**… button to open the border editor dialog box.
8. Make sure that the box labeled **Visible** is checked.
9. Select the cursor width you want from the **Width** spin edit box.
10. To change the cursor color, select the **Color...** button to open the color selection dialog box.
11. Click on the cursor color you want and then click **OK** to close the color selection dialog box.
12. To change the style of the cursor, select from the **Style** drop-down list.
13. Click **OK** to close the border editor dialog box.
14. Click the **Close** button to finish.

Moving the Series Marks

To use the **Drag Marks** tool discussed in this section, the series marks must be visible. To make the marks visible, right-click on the chart to open the **Edit** menu, then click **Edit Options**. Select the **Series** tab and then the **Marks** sub-tab. Finally, ensure the box labeled **Visible** is checked.

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Drag Marks** tool if necessary. See the **Adding a Tool** section for more information.



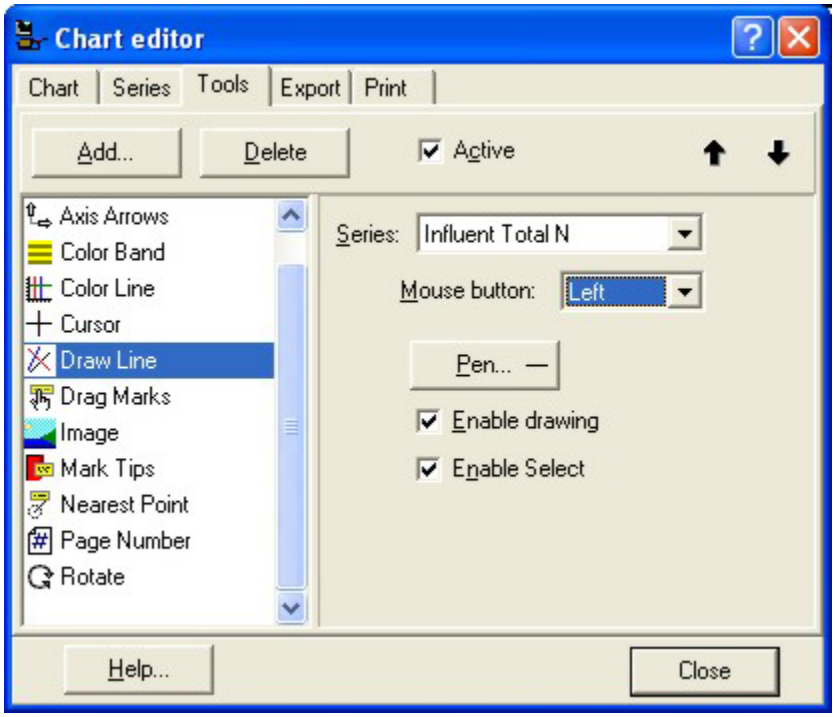
Tools drag marks dialog box

4. All series marks will be movable at this stage. To adjust only a specific series in the graph, select from the drop-down list labeled **Series**.
5. To return the marks to their original positions, use the **Reset positions** button in this dialog box.
6. Click the **Close** button to finish.
7. Adjust the position of the series marks in the graph as required by dragging.

Drawing Lines in the Chart

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.

2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Draw Line** tool if necessary. See the **Adding a Tool** section for more information.

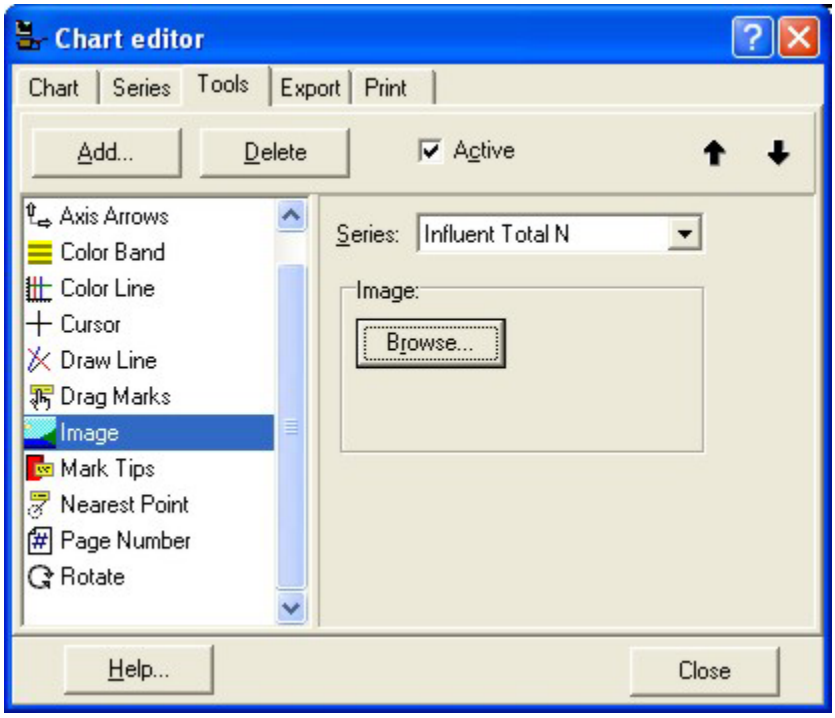


Tools draw line dialog box

4. Ensure the box labeled **Active** is checked. If this is not checked, you will be unable to draw lines and any drawn lines will be removed from the chart.
5. Ensure the box labeled **Enable drawing** is checked.
6. To change the mouse button used to draw lines in the chart, select from **Left**, **Right** or **Middle** in the **Mouse button** drop-down list.
7. To change the appearance of the lines, click the **Pen**… button to open the border editor dialog box.
8. Make sure that the box labeled **Visible** is checked.
9. Select the line width you want from the **Width** spin edit box.
10. To change the line color, select the **Color...** button to open the color selection dialog box.
11. Click on the line color you want and then click **OK** to close the color selection dialog box.
12. To change the style of the line, select from the **Style** drop-down list.
13. Click **OK** to close the border editor dialog box.
14. If you wish to move the drawn lines by dragging with the mouse, ensure the box labeled **Enable Select** is checked.
15. Click the **Close** button to finish.

Using the Image Tool

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Image** tool if necessary. See the **Adding a Tool** section for more information.

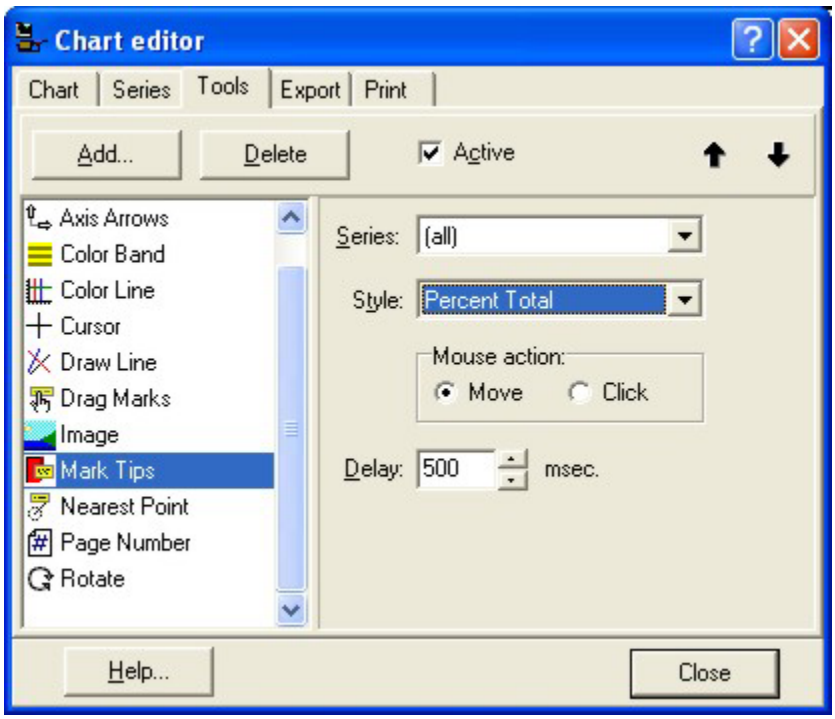


Tools image dialog box

4. To add an image, select the **Browse...** button to open the file selection dialog box.
5. Choose the graphic image you want and then click **OK** to close the file selection dialog box.
6. Click the **Close** button to finish.

Displaying Series Labels by Pointing

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Mark Tips** tool if necessary. See the **Adding a Tool** section for more information.

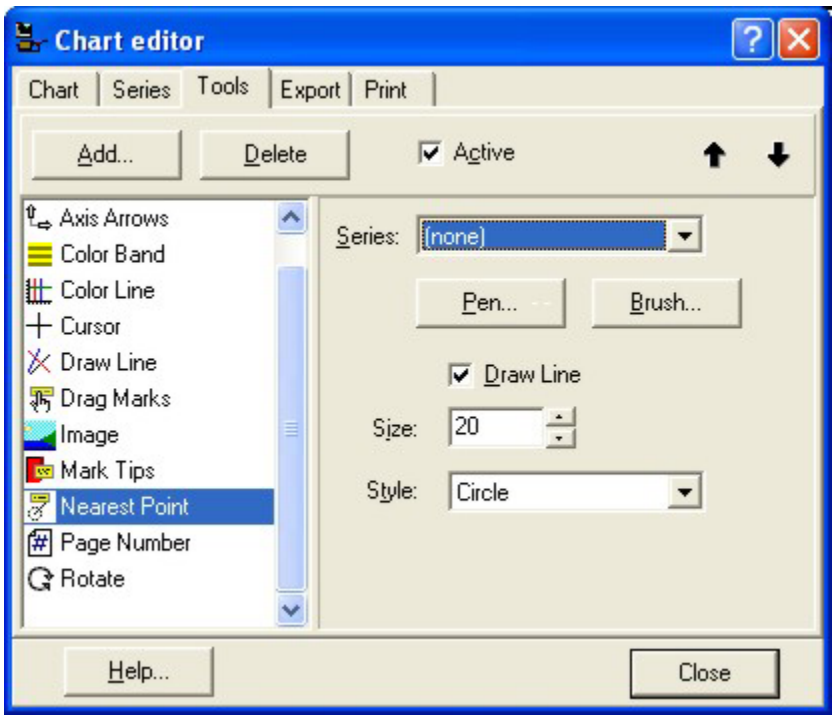


Tools mark tips dialog box

4. Ensure the box labeled **Active** is checked.
5. Select the series you wish to display mark tips for using the drop-down list labeled **Series**.
6. Choose the content of the mark tip (eg. **Label**, **Label and Percent**, **Percent Total**, etc.) from the drop-down list labeled **Style**.
7. Choose the mouse action (**Move** or **Click**) that will cause the mark tip to display using the radio buttons in the section labeled **Mouse action**.
8. To adjust the time delay between the mouse action and the display of the mark tip, increase or decrease the value in the **Delay** spin edit box.
9. Click the **Close** button to finish.

Emphasizing Data with a Pointer

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Nearest Point** tool if necessary. See the **Adding a Tool** section for more information.



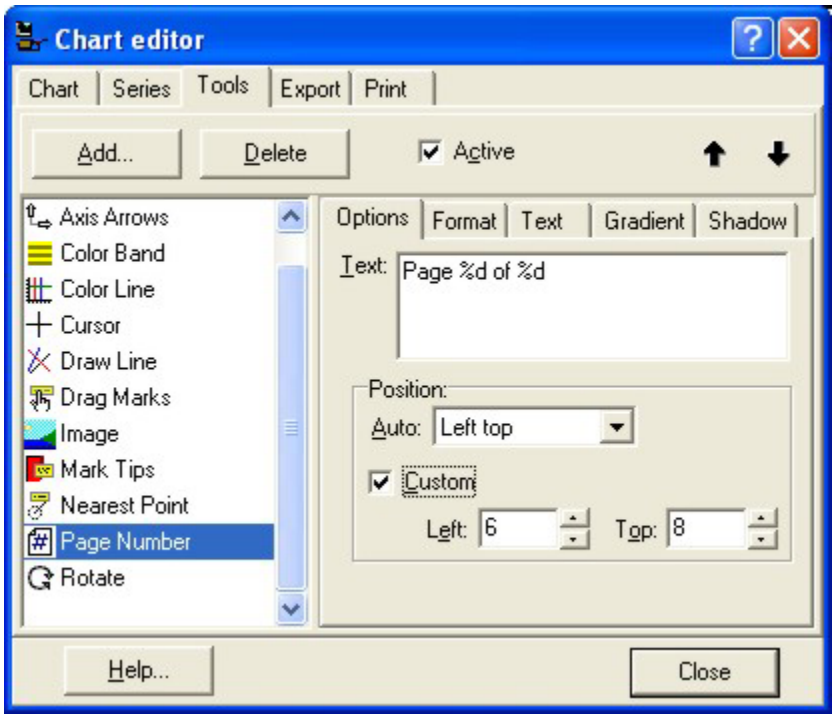
Tools nearest point dialog box

4. To choose the series you wish to point at, select from the drop-down list labeled **Series**.
5. To change the appearance of the pointer, click the **Pen**… button to open the border editor dialog box.
6. Make sure that the box labeled **Visible** is checked.
7. Select the line width you want from the **Width** spin edit box.
8. To change the line color, select the **Color...** button to open the color selection dialog box.
9. Click on the line color you want and then click **OK** to close the color selection dialog box.
10. To change the style of the line, select from the **Style** drop-down list.
11. Click **OK** to close the border editor dialog box.
12. To change the pattern of the pointer, click the **Brush…** button to open the pattern selection dialog box.
13. Click to select the pattern you require from the pattern list.
14. To change the pattern color, select the **Color...** button to open the color selection dialog box.
15. Click on the pattern color you want and then click **OK** to close the color selection dialog box.
16. Click **OK** to close the pattern selection dialog box.
17. If a line from the mouse to the data point is desired, ensure the box labeled **Draw Line** is checked.

18. To adjust the size of the pointer, increase or decrease the value in the **Size** spin edit box.
19. To change the style of the pointer, choose from **Circle**, **Rectangle**, **Diamond** or **None** from the **Style** drop-down list.
20. Click the **Close** button to finish.

Custom Page Numbering

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Page Number** tool if necessary. See the **Adding a Tool** section for more information.



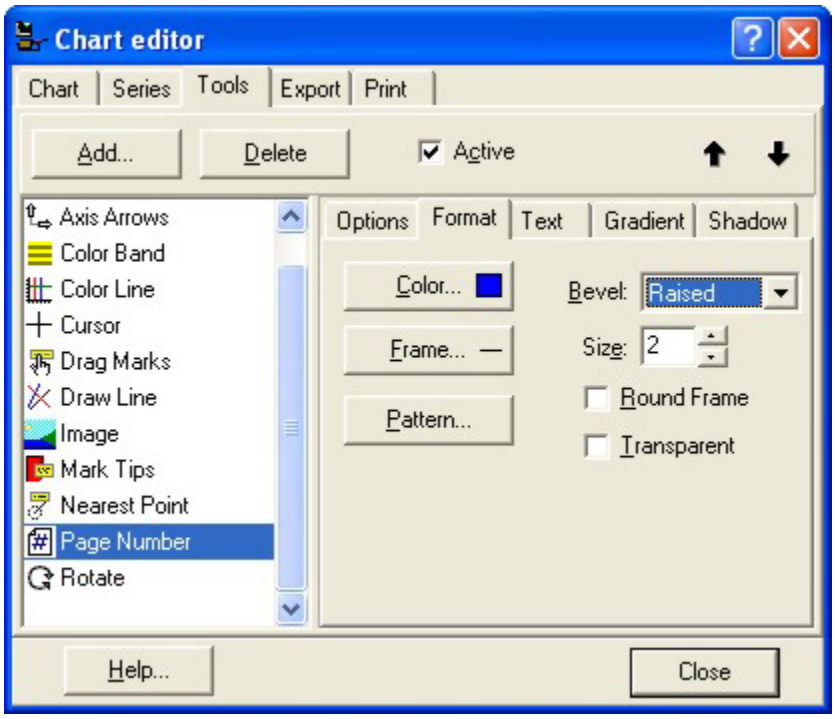
Tools page number options dialog box

4. Ensure the check box labeled **Active** is checked.
5. In the section labeled **Text**, edit the page number as required, pressing the Enter key each time you want to start a new line. The page number box in the graph will automatically adjust to the width and height of the lines you type.
6. To change the position of the page number within the graph, use the **Auto** drop-down list in the **Position** section to choose a location (**Left top**, **Left bottom**, **Right top**, **Right bottom**).
7. To set a custom location for the page number, check the box labeled **Custom**, and adjust the position using the **Left** and **Top** spin edit boxes.

8. Click the **Close** button to finish.

Changing the Page Number Format

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Page Number** tool if necessary. See the **Adding a Tool** section for more information.
4. Click the **Format** sub-tab.



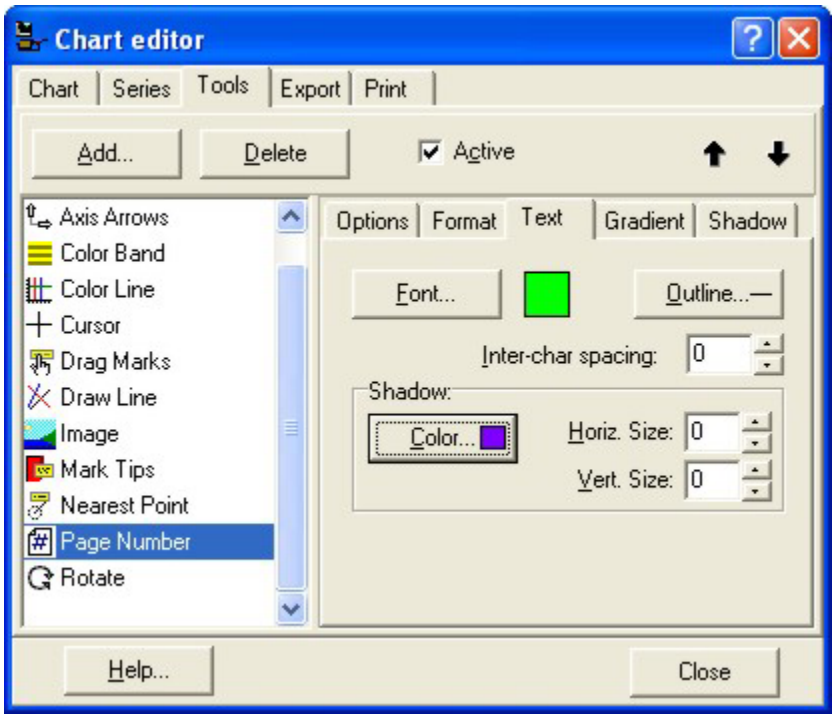
Tools page number format dialog box

5. Select the **Color...** button to open the color selection dialog box.
6. Click on the background color you want and then click **OK** to close the color selection dialog box.
7. Click **OK** to close the border color editor.
8. To adjust the appearance of the page number frame, select the **Frame...** button to open the color selection dialog box.
9. Make sure that the box labeled **Visible** is checked.
10. Select the frame style you want from the **Style** drop-down list.
11. Select the frame width you want from the **Width** spin edit box.
12. To change the frame color, select the **Color...** button to open the color selection dialog box.
13. Click on the frame color you want and then click **OK** to close the color selection dialog box.

14. Click **OK** to close the border editor.
15. If a rounded box is required, check the box labeled **Round Frame**.
16. If a three-dimensional effect is required, select **Raised** or **Lowered** from the **Bevel** drop-down list. The depth of this effect can be adjusted with the **Size** spin edit box.
17. If a background pattern is required, select the **Pattern…** button to open the pattern color editor dialog box.
18. Select the fill style you want from the list box.
19. To change the fill color, select the **Color...** button to open the color selection dialog box.
20. Click on the color you want and then click **OK** to close the color selection dialog box.
21. Click **OK** to close the pattern color editor.
22. Click the **Close** button to finish.

Changing the Page Number Text

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Page Number** tool if necessary. See the **Adding a Tool** section for more information.
4. Click the **Text** sub-tab.

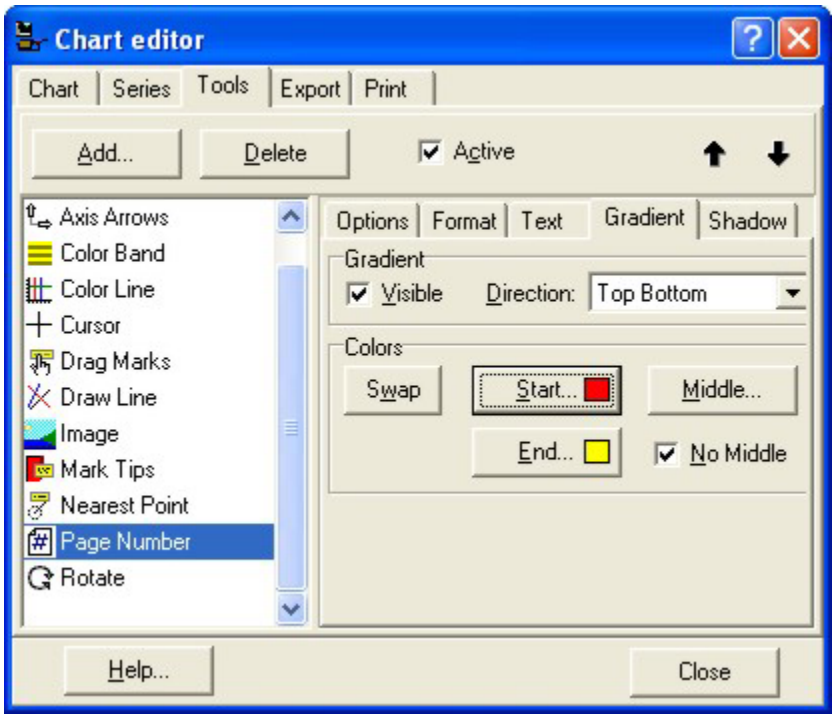


Tools page number text dialog box

5. Click the **Font…** button to display the Font Properties dialog box and select the **Font** (e.g. Arial, Times New Roman, etc), the **Font Style** (e.g. Italic, Bold, etc.), **Size**, and **Color** until the **Sample Text** has the appearance you want.
6. Click the **OK** button to close the Font Properties dialog box.
7. Click the **Shadow…** button to display the Color dialog box and select from the **Basic Colors** to add a shadow to your text if desired.
8. Click the **OK** button to close the Color dialog box.
9. Click the spin edit boxes to adjust the horizontal and vertical aspects of the shadow or enter a value from the keyboard.
10. Click the **Inter-char spacing** spin edit box to adjust the character spacing or enter a value from the keyboard.
11. Click the **Close** button to finish.

Changing the Page Number Gradient

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Page Number** tool if necessary. See the **Adding a Tool** section for more information.
4. Click the **Gradient** sub-tab.

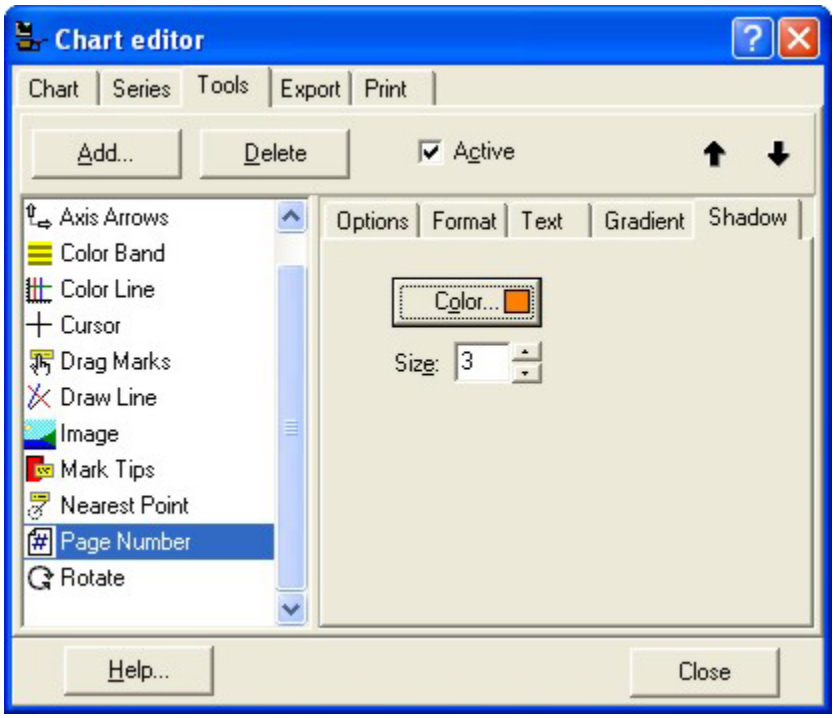


Tools page number gradient dialog box

5. Ensure the **Visible** box is checked.
6. Select a direction (eg. Left Right, Top Bottom) for the gradient from the **Direction** drop-down list.
7. In the **Colors** section of the dialog box, use the **Start**, **Middle** and **End** buttons to select the colors for the gradient. If no middle color is required, check the **No Middle** box.
8. To reverse the start and end colors, use the **Swap** button.
9. Click the **Close** button to finish.

Changing the Page Number Shadow

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Page Number** tool if necessary. See the **Adding a Tool** section for more information.
4. Click the **Shadow** sub-tab.

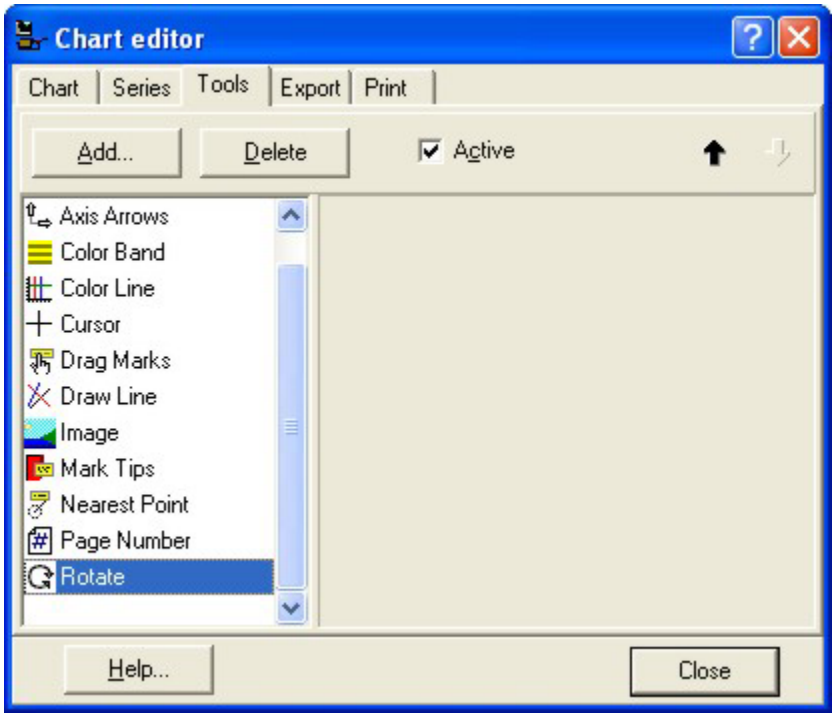


Tools page number shadow dialog box

5. Click the **Color** button to display the Color dialog box and select from the **Basic Colors** to add a shadow to your text if desired.
6. Click the **OK** button to close the Color dialog box.
7. Click the **Size** spin edit box to adjust the depth of the shadow or enter a value from the keyboard.
8. Click the **Close** button to finish.

Using the Rotate Tool

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Click the **Add…** button to access the tools list and **Add** the **Rotate** tool if necessary. See the **Adding a Tool** section for more information.



Tools rotate dialog box

4. Ensure the box labeled **Active** is checked.
5. Click the **Close** button to finish.
6. To rotate the graph freehand, press and hold the left mouse button and drag the graph as required.

Disabling a Chart Tool

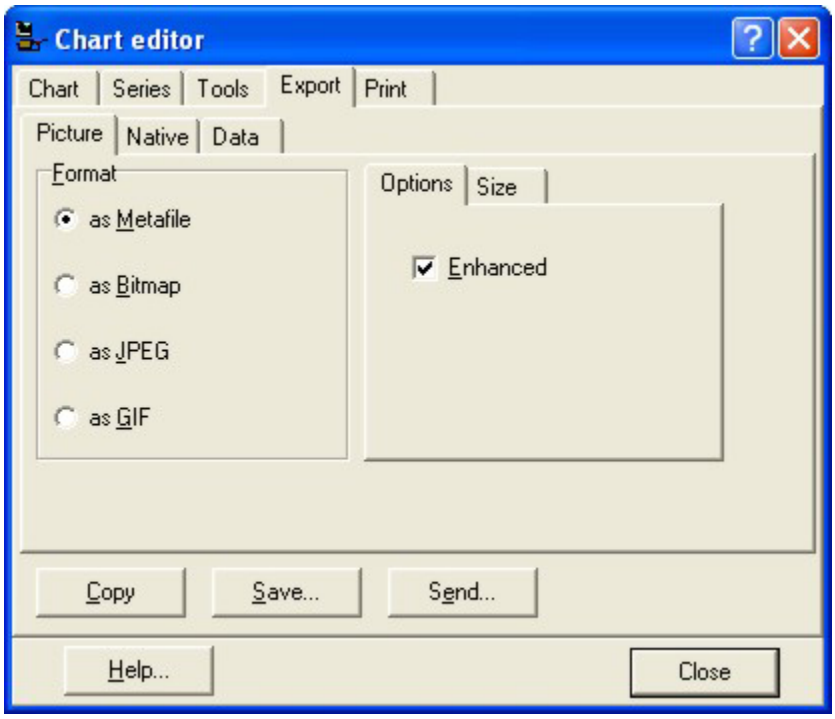
1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Tools** tab.
3. Choose the tool you want to disable from the list.
4. Remove the check from the box labeled **Active**.
5. Click the **Close** button to finish.

Export Options

This section contains procedures for exporting your chart to other programs and includes a discussion of the different file types that are available, as well as sending your chart through email.

Selecting the File Format Settings

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Export** tab.



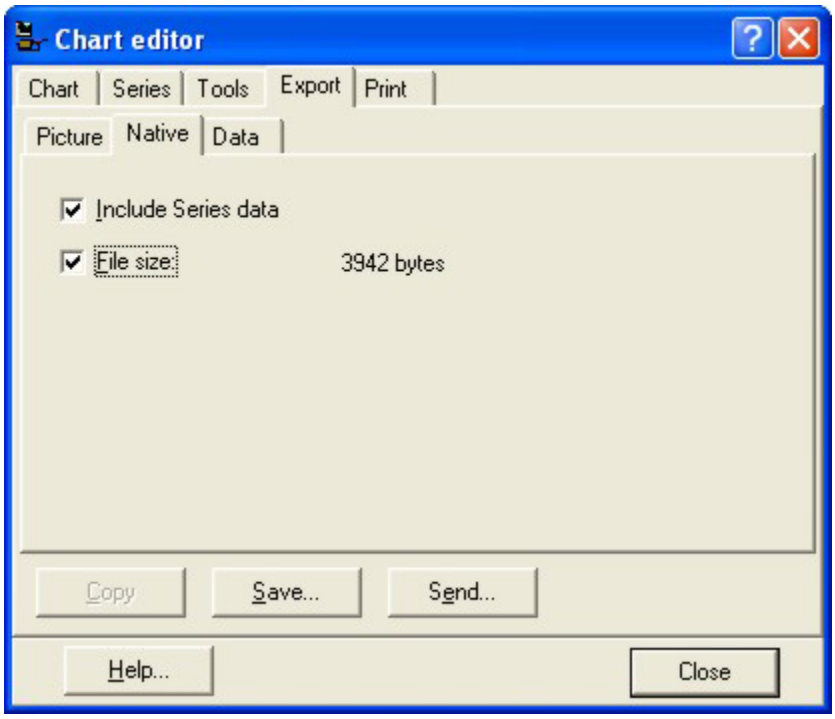
Export picture dialog box

3. Select the **Picture** sub-tab.
4. Choose the file format you want to use (**Metafile**, **Bitmap**, **GIF** or **JPEG**) using the radio buttons in the **Format** section.
5. Adjust specific file format options for export by selecting the **Options** sub-tab and applying the desired settings. Note that the options available vary with the file format selected.
6. Adjust the width and height of the exported chart by selecting the **Size** sub-tab and increasing or decreasing the values in the **Width** and **Height** spin edit boxes. To keep the chart size in perspective, check the box labeled **Keep aspect ratio**.

Including Series Data

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Export** tab.

3. Select the **Native** sub-tab.

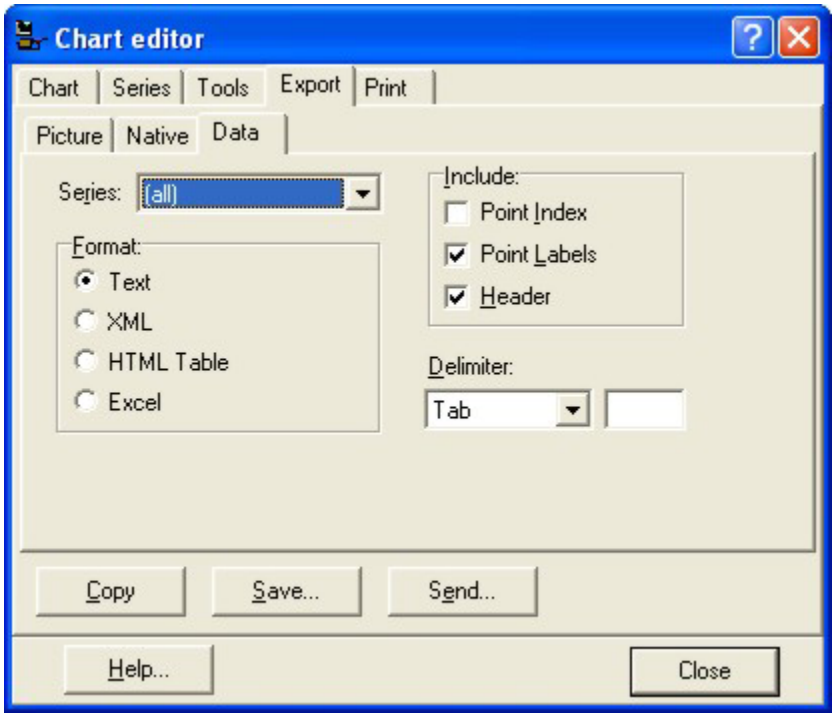


Export native dialog box

4. To include the series data with the chart for export, check the box labeled **Include Series data**.
5. To determine the size of the file to be exported, check the box labeled **File Size**.

Selecting the Data Series Format Settings

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Export** tab.
3. Select the **Data** sub-tab.



Export data dialog box

4. Select the data series you wish to export using the drop-down list labeled **Series**.
5. Choose the file format for the data series (**Text**, **XML**, **HTML-Table** or **Excel**) using the radio buttons in the **Format** section.
6. Include the point index, point labels and header information by checking the boxes in the section labeled **Include**.

Exporting an Email Attachment

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Export** tab.
3. Select the export settings required using the **Picture**, **Native** and **Data** sub-tabs.
4. Click the **Send…** button to open the personal folders dialog box.
5. Enter your password and press **OK**.
6. Your Email program will create a new email message and attach the file. Complete the message and send it. Close your Email program when finished.

Exporting to a File

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Export** tab.

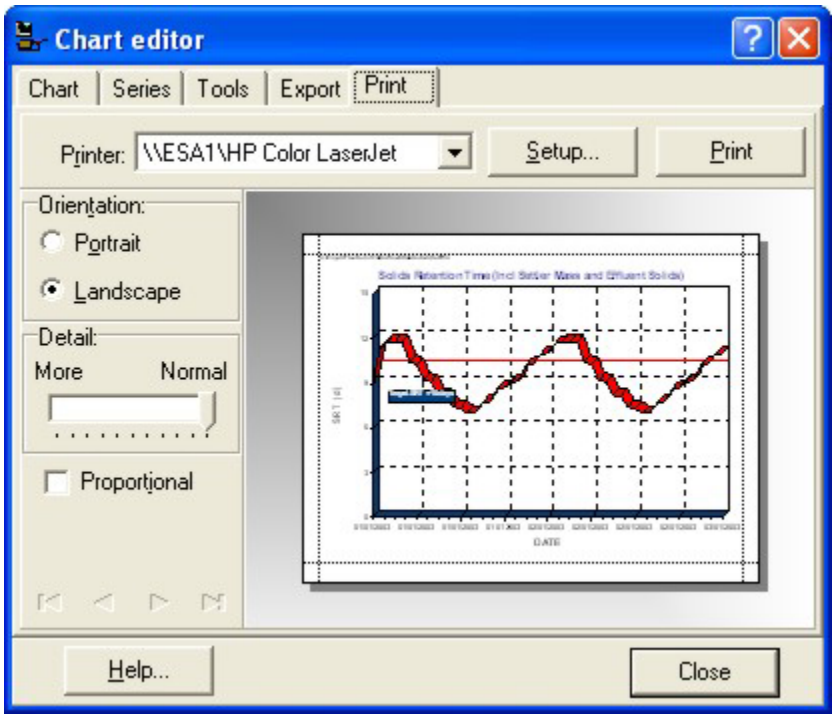
3. Select the export settings required using the **Picture**, **Native** and **Data** sub-tabs.
4. Click the **Save…** button to open the **Save As** dialog box.
5. Enter a filename for the file and choose the storage location.
6. Click the **Save** button.
7. Click the **Close** button to finish.

Print Options

This section contains procedures for printing your charts, including options for printer setup, printer selection and page setup.

Print Preview the Chart

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Print** tab.



Print preview dialog box

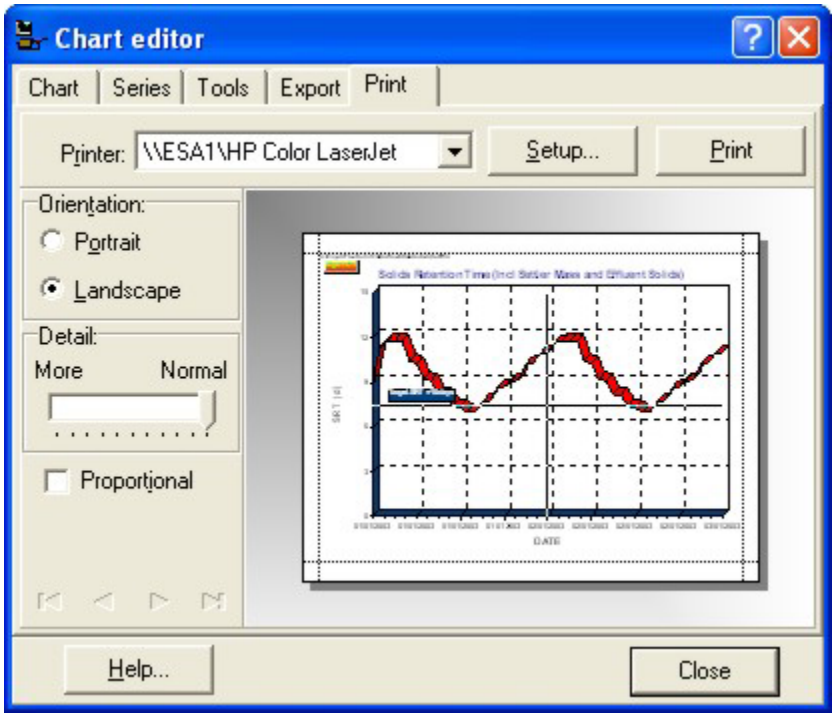
3. This dialog box can be used to:
   - Select your printer from the **Printer** drop-down list.
   - Choose your paper orientation (**Portrait** or **Landscape**) from the **Orientation** radio buttons.
   - Adjust the margins by dragging the margin lines on the previewed page.

- Dragging the pointer in the section called **Detail** between **More** and **Normal** will scale the chart text sizes. Smaller text (More detail) generally looks better when printing.
- Change your printer setup options (such as paper size, paper source) using the **Setup…** button.
- Print the previewed page using the **Print** button.

4. Click the **Close** button to finish.

Changing the Paper Orientation

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Print** tab.

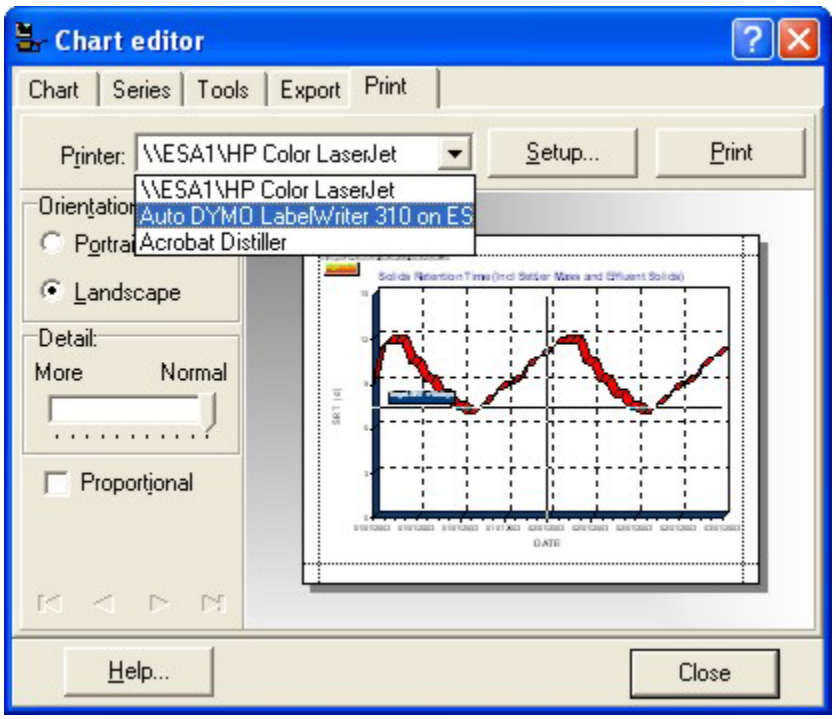


Print dialog box

3. Change between **Landscape** and **Portrait** orientation using the radio buttons in the **Orientation** section.
4. Click the **Close** or **Print** button to finish.

Selecting a Printer

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Print** tab.

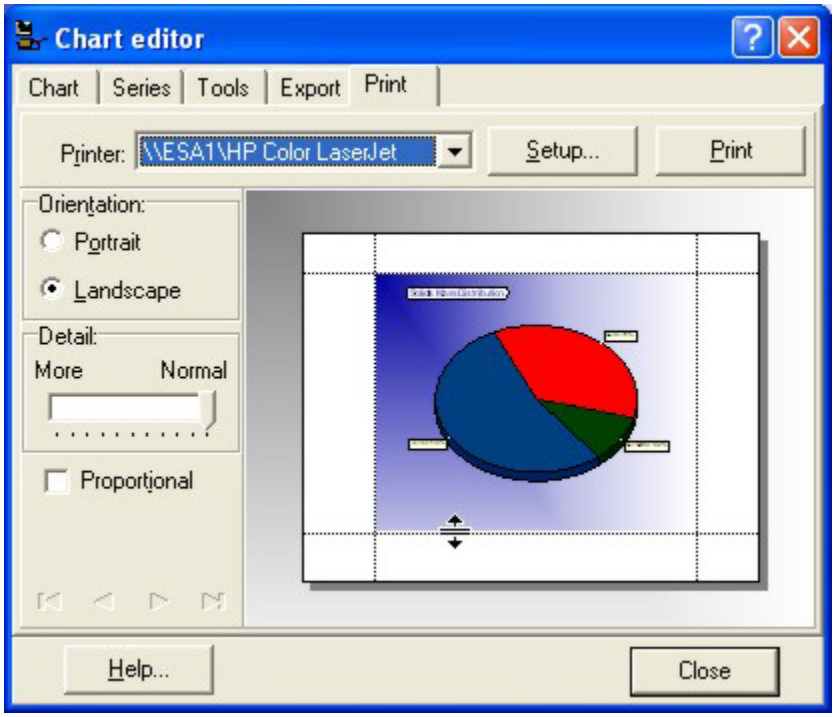


Printer selection drop-down list in the print dialog box

3. Using the drop-down list labeled **Printer**, select the printer required.
4. Click the **Print** button to finish.

Adjusting the Chart Size

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Print** tab.
3. To adjust the chart margins and resize the chart, point the mouse at the **dashed lines** in the print preview area of the print dialog box and drag them to the desired position. The print preview window will automatically display the adjusted chart

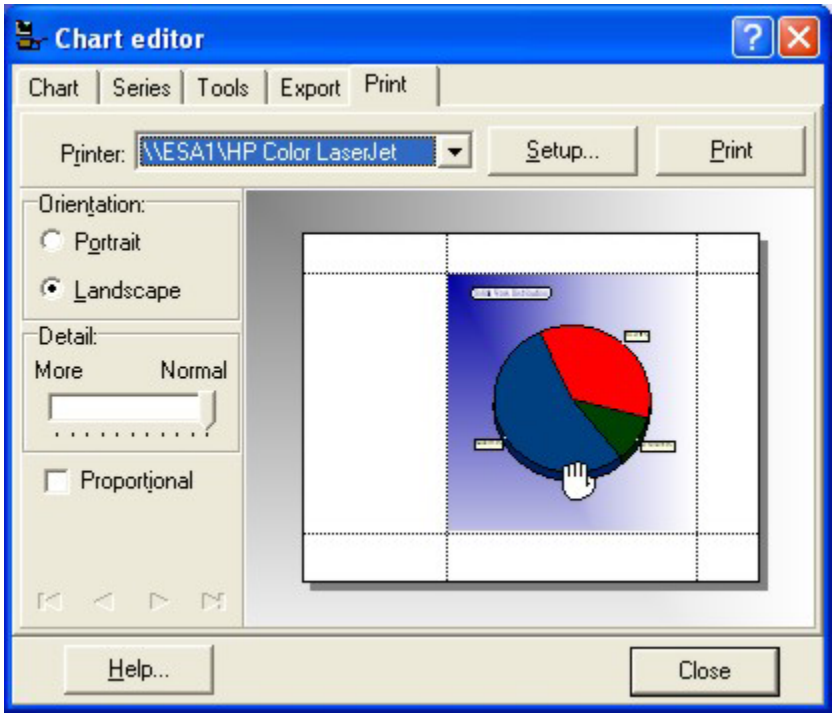


Margin adjustment in the print dialog box

4. To adjust the level of detail, drag the pointer in the section labeled **Detail** between **More** and **Normal**.
5. Click the **Close** or **Print** button to finish.

Adjusting the Chart Position

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Print** tab.
3. Position the mouse over the graph in the print preview area. The mouse pointer will change to an image of a hand.

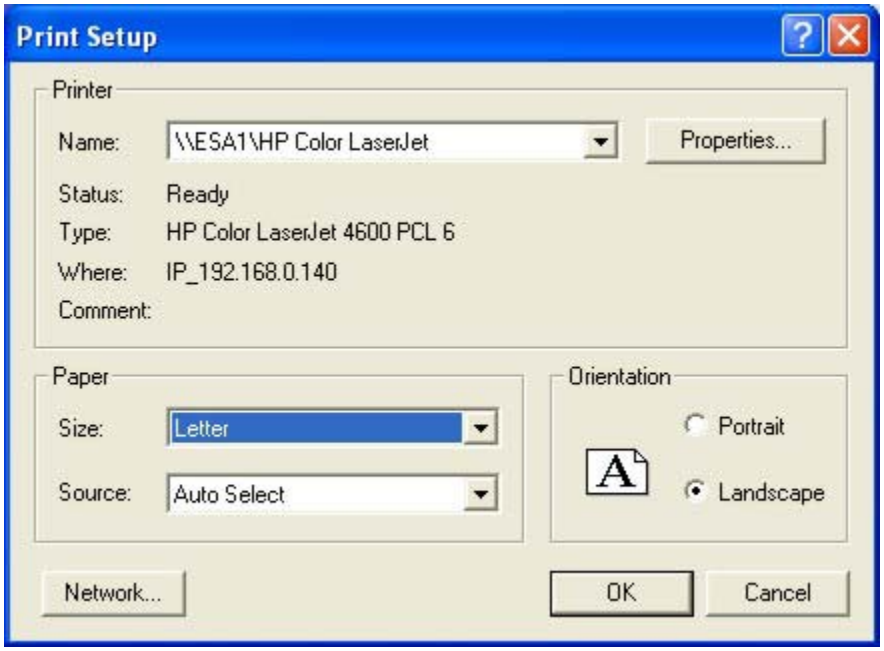


Moving the chart in the print dialog box

4. Drag the chart into the desired location.
5. Click the **Close** or Print **button** to finish.

Changing the Paper Size or Source

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Options**.
2. Select the **Print** tab.
3. Click on the **Setup…** button to open the print setup dialog box.



Print setup dialog box

4. Select the paper size (eg. **Legal**, **A4**, **Executive**) from the drop-down list labeled **Size**.
5. Select the paper feed source for the printout (eg. **Lower**, **Upper**, **Manual Feed**) from the drop-down list labeled **Source**.
6. Click the **OK** button to close the print setup dialog box.
7. Click the **Print** or **Close** button to finish.

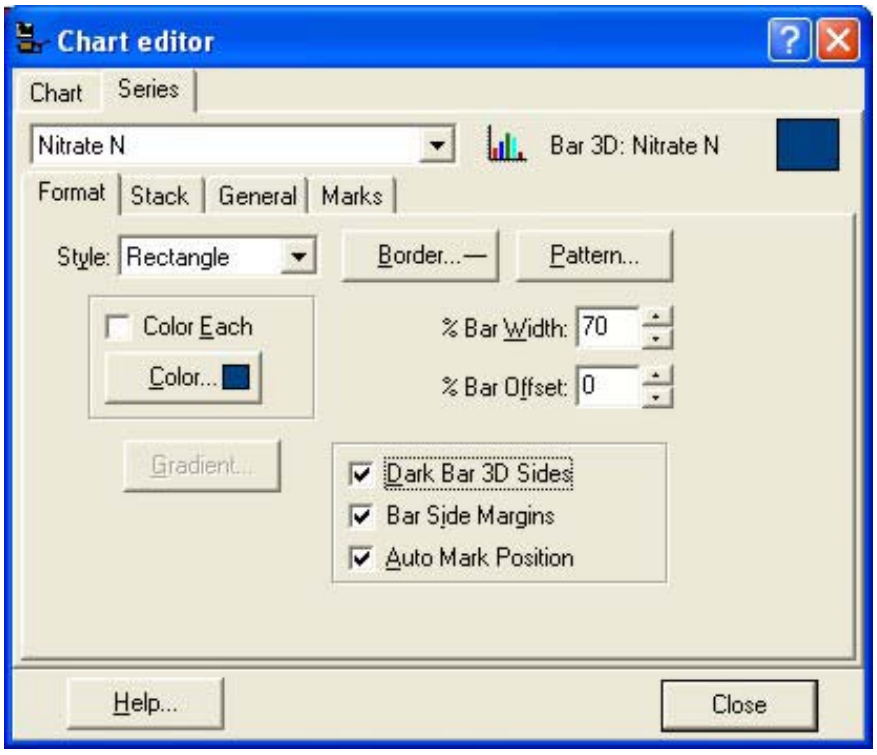
Series Formatting Procedures

Series Formatting Note

One of BioWin's helpful series formatting features is that changes you make to a series are displayed "on the fly" as you make them. This is an excellent way to learn the functionality of the series editing controls since you immediately see the results of clicking various buttons, spin edit boxes, and other dialog box controls in the background. The associations between dialog controls and their resultant changes to the series are learned much more quickly than if you have to exit the dialog or manually apply the changes before you see their effects.

Common Series Procedures

This section covers procedures that are accessible from the **Series** tab of the **Edit Series** and **Edit Options** commands.



Tab used for series format manipulation

Change a Series Style

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series** or **Edit Options** command.
2. In the **Series** list box on the **Chart** tab, select the series you want to change the style of.
3. Click the **Change...** button.
4. Select the new series style from the **Series Gallery.** If a 3D style is required, ensure the **3D** box is checked**.** Click the **OK** button when done.
5. Click the **Close** button to finish.

Note: To change the style of another series, repeat steps 2-4.

Rename a Series

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series** or **Edit Options** command.
2. In the **Series** list box on the **Chart** tab, select the series you want to rename.
3. Click the **Title...** button.
4. Enter the new name in the **Change Series Title** dialog box, and click **OK**.
5. Click the **Close** button to finish.

Note: To rename another series, repeat steps 2-4.

Delete a Series From a Chart

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series** or **Edit Options** command.
2. In the **Series** list box on the **Chart** tab, select the series you want to delete.
3. Click the **Delete** button.
4. If you are sure you want to delete this series, click **Yes** in the confirmation box.
5. Click the **Close** button to finish.

Note: To delete another series, repeat steps 2-4. You can also delete series from the **Edit Series List** dialog box – if you have many series this may in fact be quicker.

Add/Remove a Series in the Legend

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series** or **Edit Options** command.
2. Click the **Series** sub-tab.
3. Click the **General** sub-tab.
4. Add or remove the check from the box labeled **Show in Legend**.

5. Click the **Close** button to finish.

Note: To add/remove another series, repeat steps 2-4.

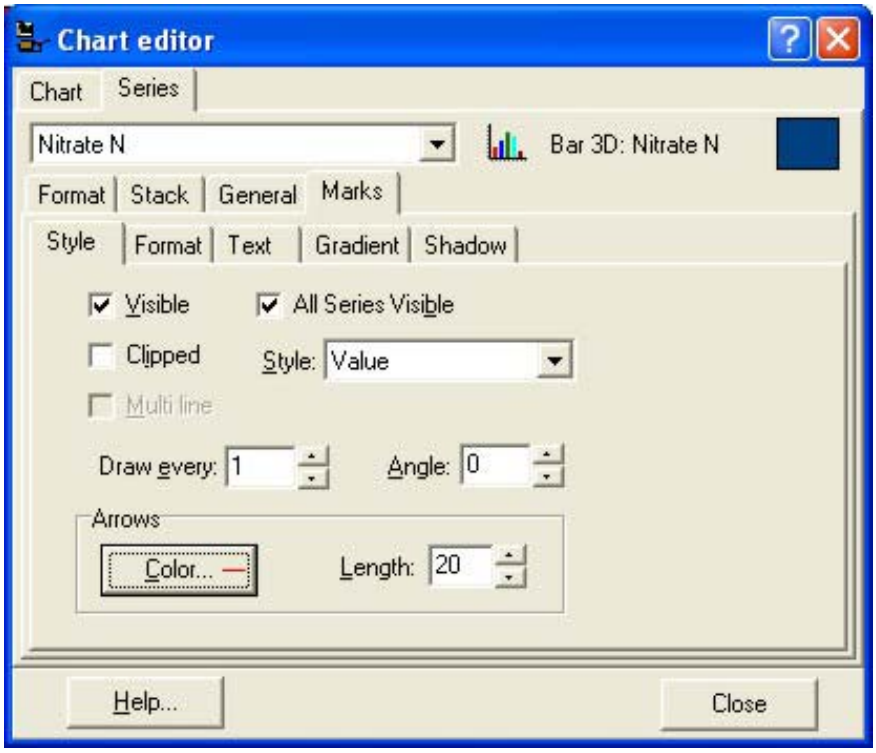
Control the Axis that a Series is Plotted On

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series** or **Edit Options** command.
2. In the **Series** list box on the **Chart** tab, select the series you want to control.
3. Click the **Series** sub-tab.
4. Click the **General** sub-tab.
5. Choose the position that you want the series plotted on (**Top**, **Bottom** or **Top and Bottom**) from the drop-down list in the section labeled **Horizontal Axis.**
6. If you want this axis to have date/time formats, check the box labeled **Date Time**.
7. In the **Vertical Axis** drop-down list select the vertical axis (**Left**, **Right** or **Left and Right**) that you want the series to be plotted on.
8. If you want this axis to have date/time formats, check the box labeled **Date Time**.
9. Click the **Close** button to finish.

Note: To change the axis for another series, repeat steps 2-8.

Series Labeling Procedures

This section outlines the various methods for changing the appearance of series labels, known as *marks*, including border, color, font, and style.



Dialog box used for manipulating series marks

Adding Labels to a Chart

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series** or **Edit Options** command.
2. In the **Series** list box on the **Chart** tab, select the series you want to control.
3. Click the **Marks** sub-tab.
4. Make sure that the box labeled **Visible** is checked
5. Click the **Close** button to finish.

Series Label Styles

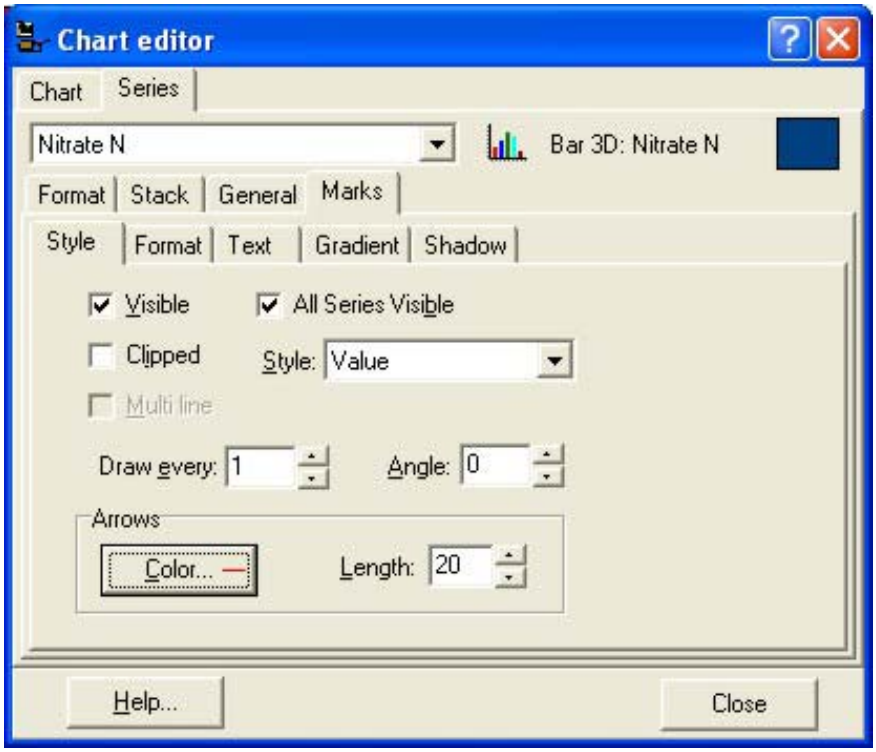
This section gives an outline and examples of the various series label styles that are available in BioWin.

Style	Example
Value	1234
Percent	12 %
Label	Mass
Label & Percent	Mass 25 %
Label & Value	Mass 1234
Legend	Depends on Legend Text Style
Percent Total	12 % of 1234
Label & Percent Total	Mass 12 % of 1234

Xvalue 4321

Change the Series Label Style

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series** or **Edit Options** command.
2. In the **Series** drop-down list on the **Series** tab, select the series you want to control.
3. Click the **Marks** sub-tab.



Dialog box used to manipulate the style of chart marks

4. Make sure that the box labeled **Visible** is checked.
5. In the **Style** drop-down list select the label style that you want for the series.
6. Click the **Close** button to finish.

Note: To change the label style for another series, repeat steps 2-5.

Change the Series Label Number Format

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series** or **Edit Options** command.
2. In the **Series** drop-down list on the **Series** tab, select the series you want to control.
3. Click the **Marks** sub-tab.

4. Click the **General** sub-tab.

In the **Formats** group, change the **Values** or **Percents** formatting to the style you want. Refer to the **BioWin Number Formats** topic below for more information.

5. Click **OK** to finish.

BioWin Number Formats

This section contains an overview of the various number formats that are available in BioWin. Although this information is applicable elsewhere, it has been included here since it will most likely be needed in relation to chart labeling.

Specifier	Description
0	Digit placeholder. If the value being formatted has a digit in the position where the '0' appears in the format string, then that digit is copied to the output string. Otherwise, a '0' is stored in that position in the output string.
#	Digit placeholder. If the value being formatted has a digit in the position where the '#' appears in the format string, then that digit is copied to the output string. Otherwise nothing is stored in that position in the output string.
.	Decimal point. The first '.' character in the format string determines the location of the decimal separator in the formatted value; any additional '.' characters are ignored. The actual character used as a decimal separator in the output string is specified in the Number Format of the International section of the Windows Control Panel.
,	Thousand separator. If the format string contains one or more ',' characters, the output will have thousand separators inserted between each group of three digits to the left of the decimal point. The placement and number of ',' characters in the format string does not affect the output, except to indicate that thousand separators are wanted. The actual character used as a thousand separator in the output string is specified in the Number Format of the International section of the Windows Control Panel.
E+	Scientific notation. If any of the strings 'E+', 'E-', 'e+', or 'e-' are contained in the format string, the number is formatted using scientific notation. A group of up to four '0' characters can immediately follow the 'E+', 'E-', 'e+', or 'e-' to determine the minimum number of digits in the exponent. The 'E+' or 'e+' formats cause a plus sign to be output for positive exponents and a minus sign to be output for negative exponents. The 'E-' and 'e-' formats output a sign character only for negative exponents.
'xx'/"xx"	Characters enclosed in single or double quotes are output as-is, and do not affect formatting.
;	Separates sections for positive, negative, and zero numbers in the format string.

The locations of the leftmost '0' before the decimal point in the format string and the rightmost '0' after the decimal point in the format string determine the range of digits that are always present in the output string.

The number being formatted is always rounded to as many decimal places as there are digit placeholders ('0' or '#') to the right of the decimal point. If the format string contains no decimal point, the value being formatted is rounded to the nearest whole number.

If the number being formatted has more digits to the left of the decimal separator that there are digit placeholders to the left of the '.' character in the format string, the extra digits are output before the first digit placeholder.

To allow different formats for positive, negative, and zero values, the format string can contain between one and three sections separated by semicolons:

4. One section: The format string applies to all values;
5. Two sections: The first section applies to positive values and zeros, and the second section applies to negative values;
6. Three sections: The first section applies to positive values, the second applies to negative values, and the third section applies to zeros.

If the section for negative values or the section for zero values is empty, that is there is nothing between the semicolons that delimit the section, the section for the positive values is used instead.

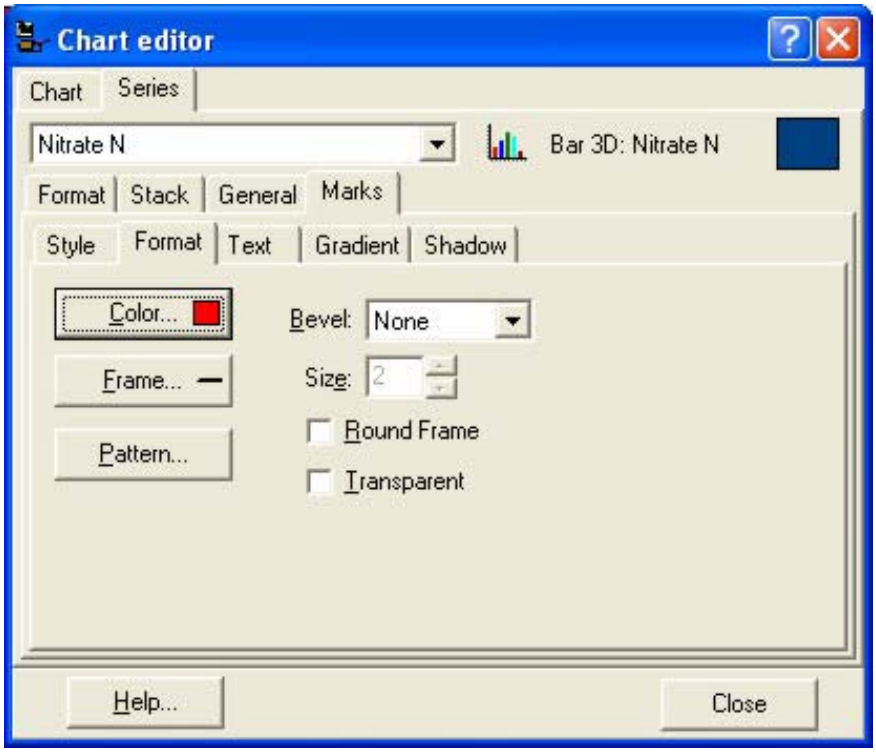
If the section for positive values is empty, or if the entire format string is empty, the value is formatted using general floating-point formatting with 15 significant digits. General floating-point formatting is also used if the value has more than 18 digits to the left of the decimal point and the format string does not specify scientific notation.

The following table shows some sample formats and the results produced when the formats are applied to different values:

Format String	1234	-1234	0.5	0
None Specified	1234	-1234	0.5	0
0	1234	-1234	1	0
0.00	1234.00	-1234.00	0.50	0.00
#.##	1234	-1234	.5	
#,##0.00	1,234.00	-1,234.00	0.50	0.00
#,##0.00;(#,##0.00)	1,234.00	(1,234.00)	0.50	0.00
#,##0.00;;Zero	1,234.00	-1,234.00	0.50	Zero
0.000E+00	1.234E+03	-1.234E+03	5.000E-01	0.000E+00
#.###E-0	1.234E3	-1.234E3	5E-1	0E0

Change the Series Label Border

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series** or **Edit Options** command.
2. In the **Series** drop-down list on the **Series** tab, select the series you want to control.
3. Click the **Marks** sub-tab.
4. Click the **Style** sub-tab.
5. Make sure that the box labeled **Visible** is checked.
6. Click the **Format** sub-tab.



Dialog box used for formatting marks

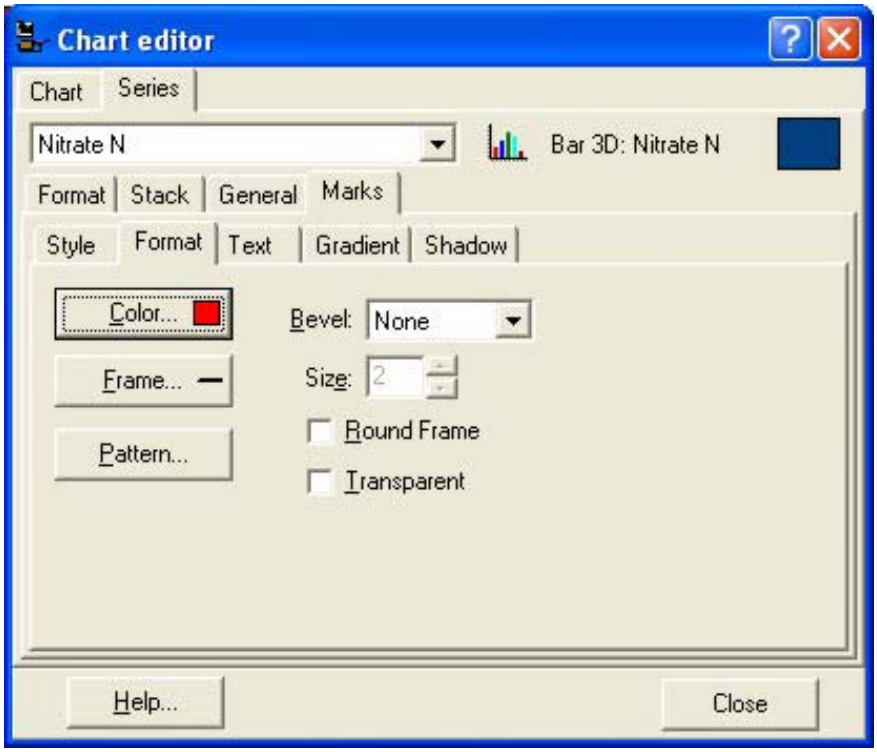
7. Click the **Frame...** button to open the border color editor.
8. Make sure that the box labeled **Visible** is checked.
9. Select the label border style you want from the **Style** drop-down list.
10. Select the label border width you want from the **Width** spin edit box.
11. To change the label border color, select the **Color...** button to open the color selection dialog box.
12. Click on the label border color you want and then click **OK** to close the color selection dialog box.
13. Click **OK** to close the border color editor.
14. Click the **Close** button to finish.

Note: To change the label border for another series, repeat steps 2-13. If you don't want series labels to extend beyond the chart axis boundaries, check the box labeled **Clip Labels**.

Change the Series Label Color

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series** or **Edit Options** command.
2. In the **Series** drop-down list on the **Series** tab, select the series you want to control.
3. Click the **Marks** sub-tab.
4. Click the **Style** sub-tab.

5. Make sure that the box labeled **Visible** is checked.
6. Click the **Format** sub-tab.



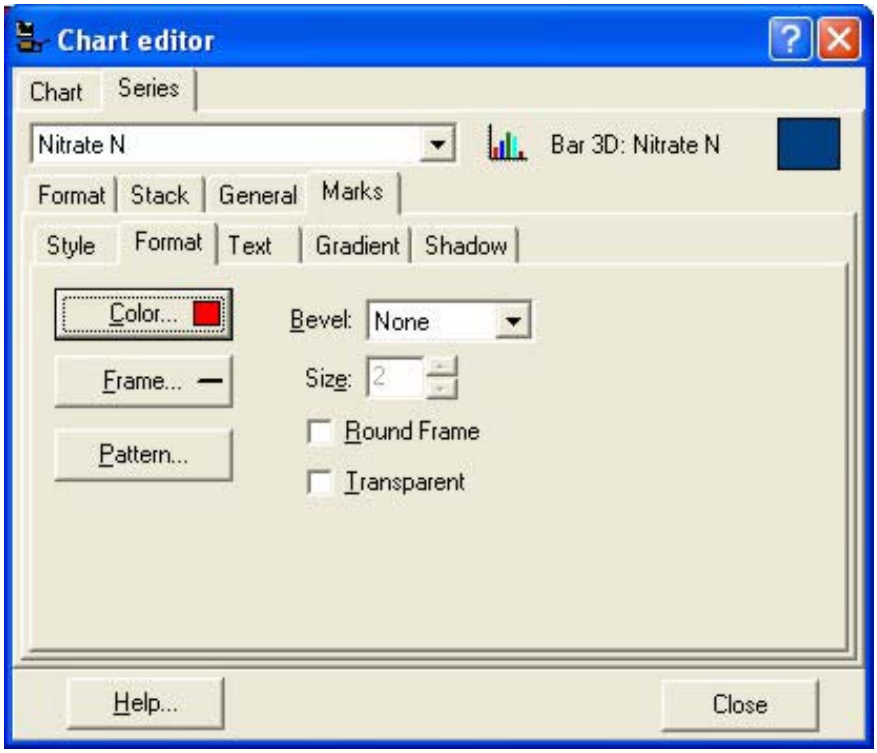
Dialog box used for formatting marks

7. Click the **Color...** button to open the color selection dialog box.
8. Click on the label color you want and then click **OK** to close the color selection dialog box.
9. Click the **Close** button to finish.

Note: To change the label color for another series, repeat steps 2-8. If you want the series labels to be clear, make the label color white.

Change the Series Label Pattern

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series** or **Edit Options** command.
2. In the **Series** drop-down list on the **Series** tab, select the series you want to control.
3. Click the **Marks** sub-tab.
4. Click the **Style** sub-tab.
5. Make sure that the box labeled **Visible** is checked.
6. Click the **Format** sub-tab.



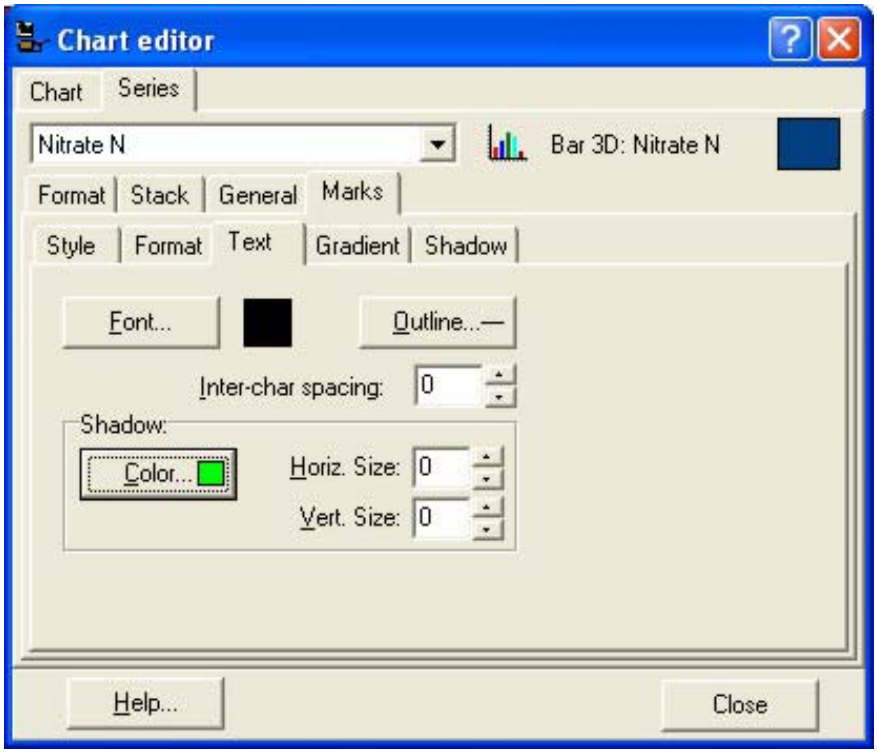
Dialog box used for formatting marks

7. Click the **Pattern...** button to open the pattern color editor dialog box.
8. Select the pattern style you want from the pattern list.
9. To change the pattern color, select the **Color...** button to open the color selection dialog box.
10. Click on the pattern color you want and then click **OK** to close the color selection dialog box.
11. Click **OK** to close the pattern color editor.
12. Click the **Close** button to finish.

Note: To change the label pattern for another series, repeat steps 2-11

Change the Series Label Font

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series** or **Edit Options** command.
2. In the **Series** drop-down list on the **Series** tab, select the series you want to control.
3. Click the **Marks** sub-tab.
4. Make sure that the box labeled **Visible** is checked.
5. Click the **Text** sub-tab.



Dialog box used for manipulating the text of series marks

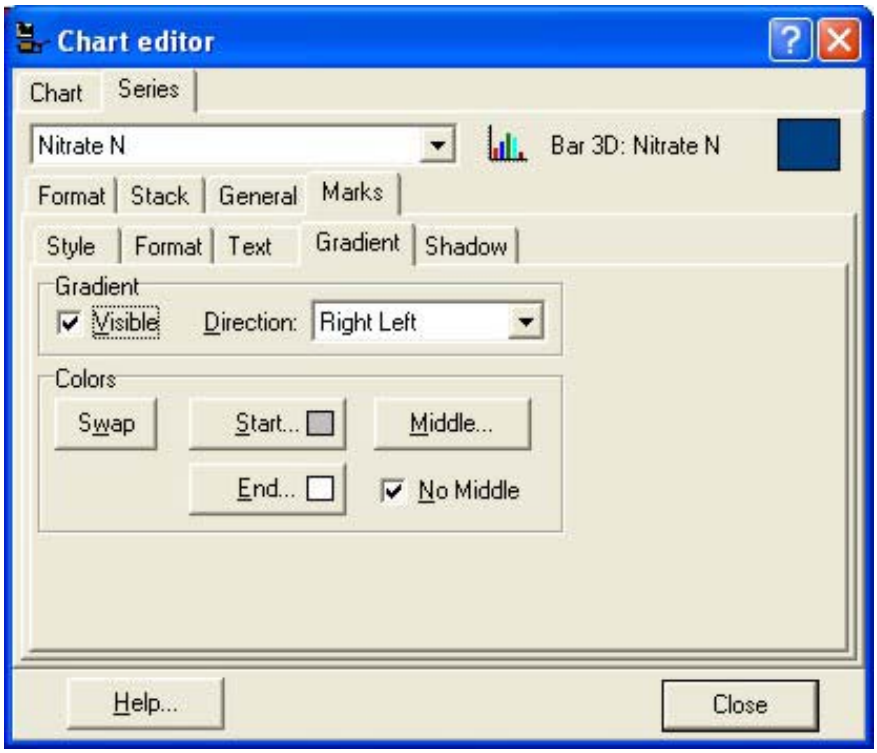
6. Click the **Font...** button to open the font properties dialog box.
7. Specify the **Font** (e.g. Arial, Times New Roman, etc), the **Font Style** (e.g. Italic, Bold, etc.), **Size**, and **Color** until the **Sample Text** has the appearance you want.
8. Click the **OK** button to close the font properties dialog box.
9. To add an outline effect to the font, click the **Outline…** button to open the border editor dialog box.
10. Choose the **Color**, **Style** and **Width** for the outline you require.
11. Click the **OK** button to close the border editor dialog box.
12. To adjust the amount of white space between the characters, increase or decrease the value in the **Inter-char spacing** spin edit box.
13. Click the **Close** button to finish.

Note: To change the label font for another series, repeat steps 2-12.

Change the Series Label Gradient

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series** or **Edit Options** command.
2. In the **Series** drop-down list on the **Series** tab, select the series you want to control.
3. Click the **Marks** sub-tab.
4. Make sure that the box labeled **Visible** is checked.

5. Click the **Gradient** sub-tab.



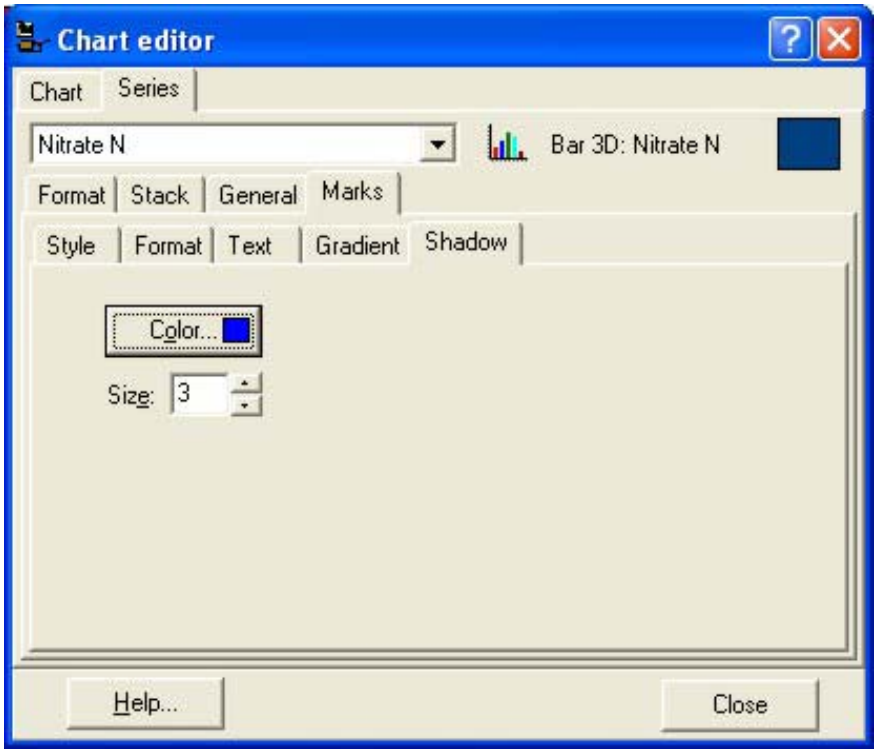
Dialog box used to add a gradient to series marks

6. Ensure the **Visible** box is checked.
7. Select a direction (eg. Left Right, Top Bottom) for the gradient from the **Direction** drop-down list.
8. In the **Colors** section of the dialog box, use the **Start**, **Middle** and **End** buttons to select the colors for the gradient. If no middle color is required, check the **No Middle** box.
9. To reverse the start and end colors, use the **Swap** button.
10. Click the **Close** button to finish.

Note: To change the label gradient for another series, repeat steps 2-9.

Change the Series Label Shadow

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series** or **Edit Options** command.
2. In the **Series** drop-down list on the **Series** tab, select the series you want to control.
3. Click the **Marks** sub-tab.
4. Make sure that the box labeled **Visible** is checked.
5. Select the **Shadow** sub-tab.



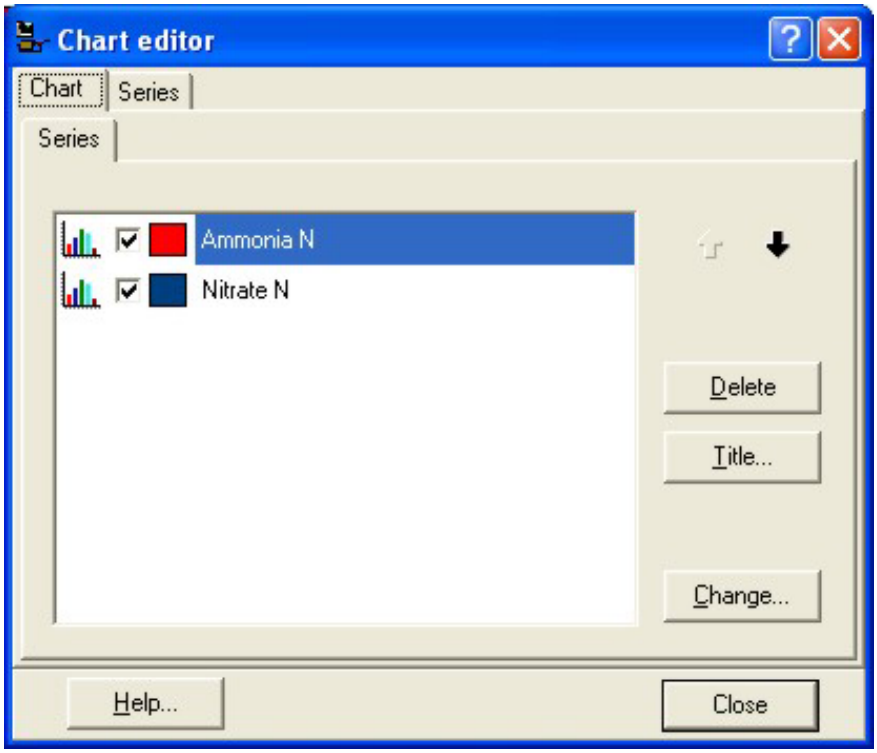
Dialog box used to add a shadow to series marks

6. Click the **Color** button to display the Color dialog box and select from the **Basic Colors** to add a shadow to your text if desired.
7. Click the **OK** button to close the Color dialog box.
8. Click the **Size** spin edit box to adjust the depth of the shadow or enter a value from the keyboard.
9. Click the **Close** button to finish.

Note: To change the label shadow for another series, repeat steps 2-8.

Edit the Series List

This section describes a number of ways to manipulate the series that are plotted on a chart. Right-clicking on a chart and selecting **Edit Series List…** from the resulting popup menu will open the following dialog box.



The Edit Series List dialog box

Hide/Show a Series

This dialog box allows you to turn the display of a series in a chart on or off. Note that this does not *remove* the series from the chart; rather it simply toggles a series' visibility. To show or hide a series in a chart, add or remove the check box beside the series name in the dialog box. If you want to *remove* a series, click on the series and press the **Delete** key on your keyboard.

Change the Series Plotting Order

You can use this dialog box to change the order in which series are plotted on a chart. To do this, click on a series name so that it is highlighted, and then click the up or down arrow. Note that when you do this, the order in which series are listed in the legend changes accordingly.

Change a Series Color

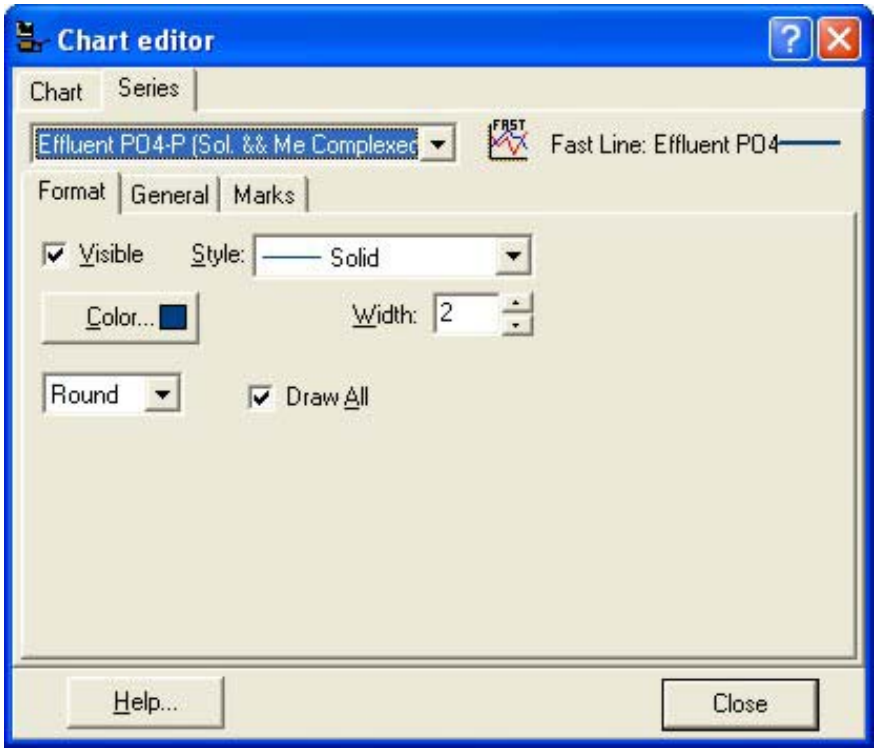
The color of each series is shown next to its name in the series list. Double-clicking on the color will open up the Color Selection dialog box. You can then use this dialog box to change the color of the series.

Fast Line Series Procedures

This section is an overview of formatting procedures that can be executed from the **Format** sub-tab of the **Edit Series** tab for a **Fast Line Series**.

Fast Line series can be used for traditional engineering X-Y plots, time series analysis, etc. In BioWin they particularly are useful for displaying dynamic

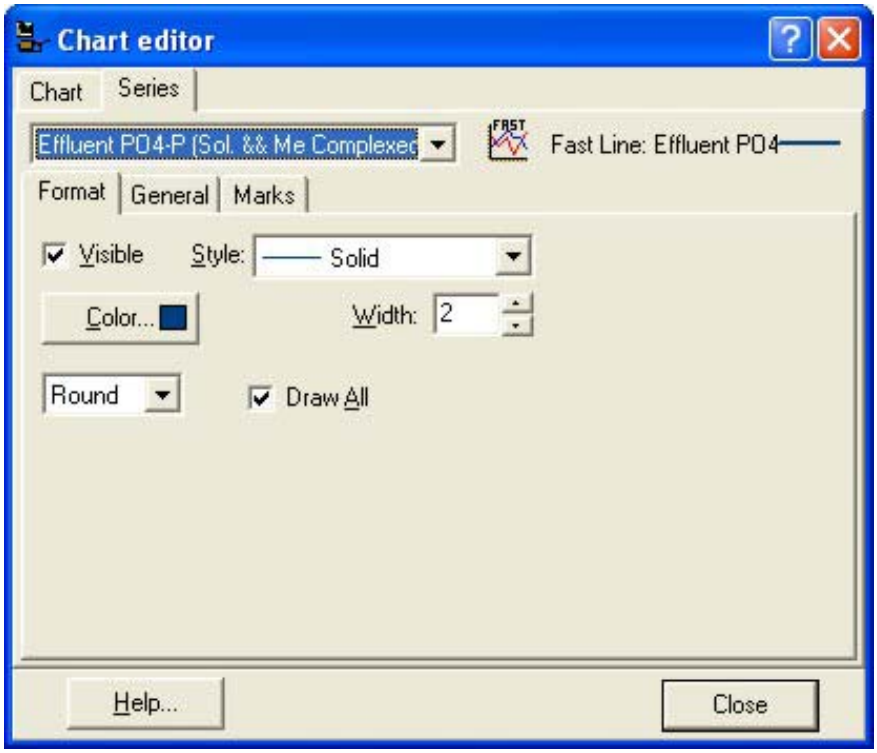
simulation results. Fast Line series have fewer display options than the regular Line series, but they draw quicker and therefore are useful if you are plotting several series at once. The Fast Line series style trades display options for speed of execution.



Fast line series formatting options

Change the Fast Line Series Appearance

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop-down list on the **Series** tab, select the fast line series you want to change the appearance of.
3. Click the **Format** sub-tab.



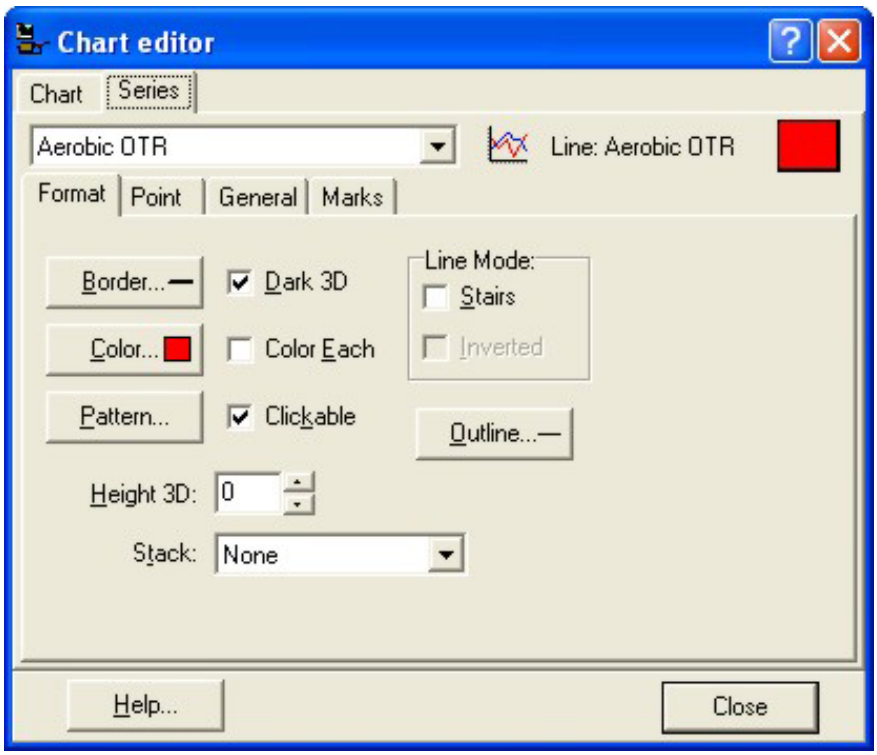
Fast line series formatting options

4. Ensure that the box labeled **Visible** is checked.
5. Change the width of the line using the **Width** spin edit box.
6. Change the line style using the **Style:** radio button group. Note that this option is not available if the line width is greater than one due to screen resolution restrictions.
7. Click the **Color...** button to open the color selection dialog box.
8. Click on the fast line series color you want and then click **OK** to close the color selection dialog box.
9. Click the **Close** button to finish.

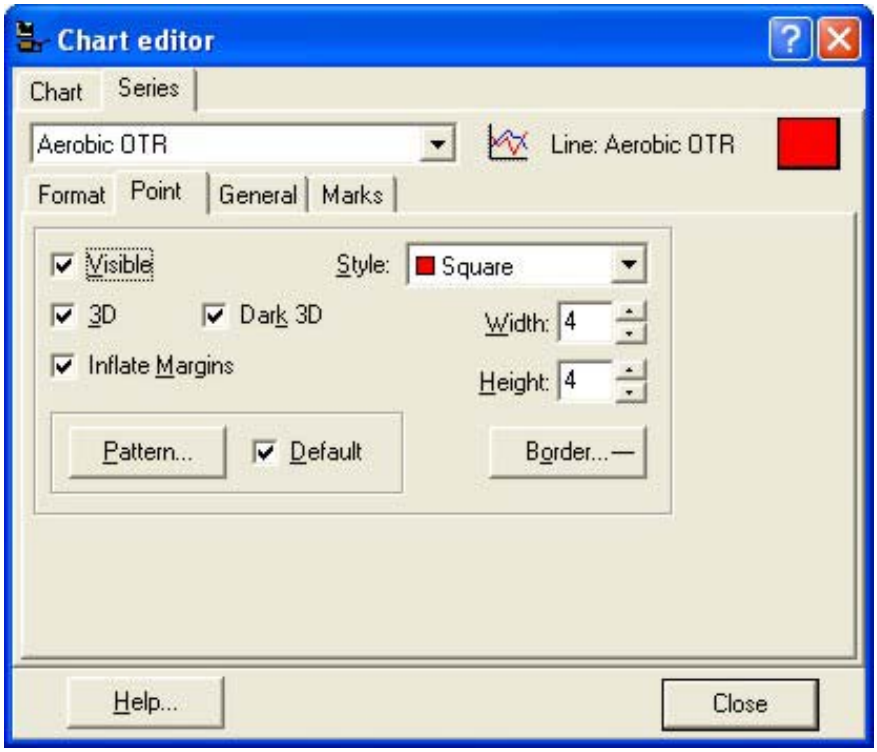
Line Series Procedures

This is an overview of formatting procedures that can be executed from the **Format** and **Point** sub-tabs of the **Edit Series** tab for a **Line Series**.

Line series can be used for traditional engineering X-Y plots, time series analysis, etc. In BioWin they particularly are useful for displaying dynamic simulation results.



Line series line formatting options

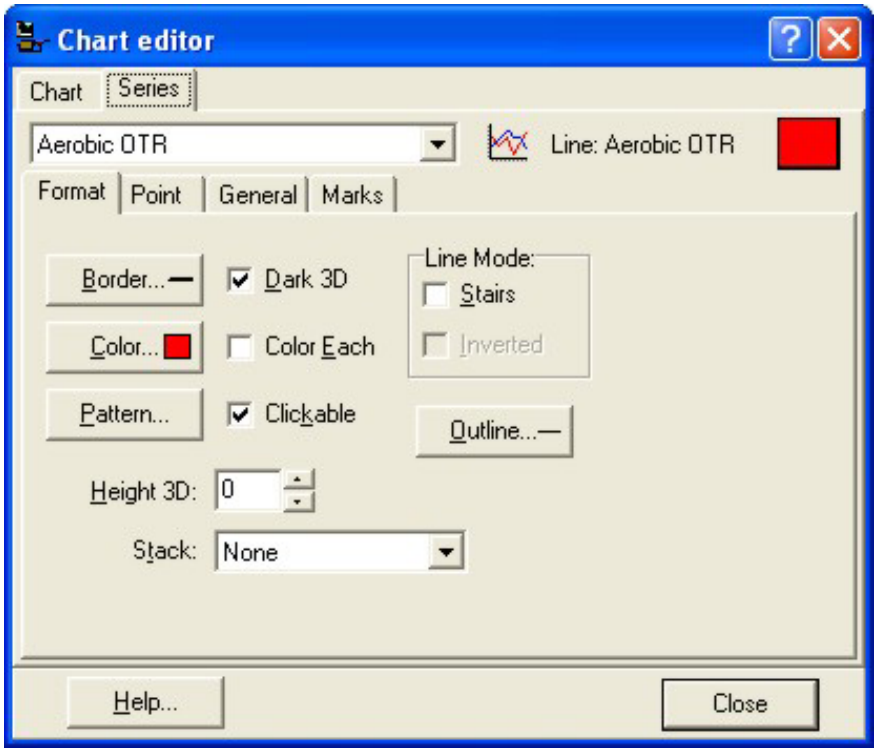


Line series point formatting options

Change the Line Series Color

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.

2. In the **Series** drop-down list on the **Series** tab, select the line series you want to change the appearance of.
3. Click the **Format** sub-tab.

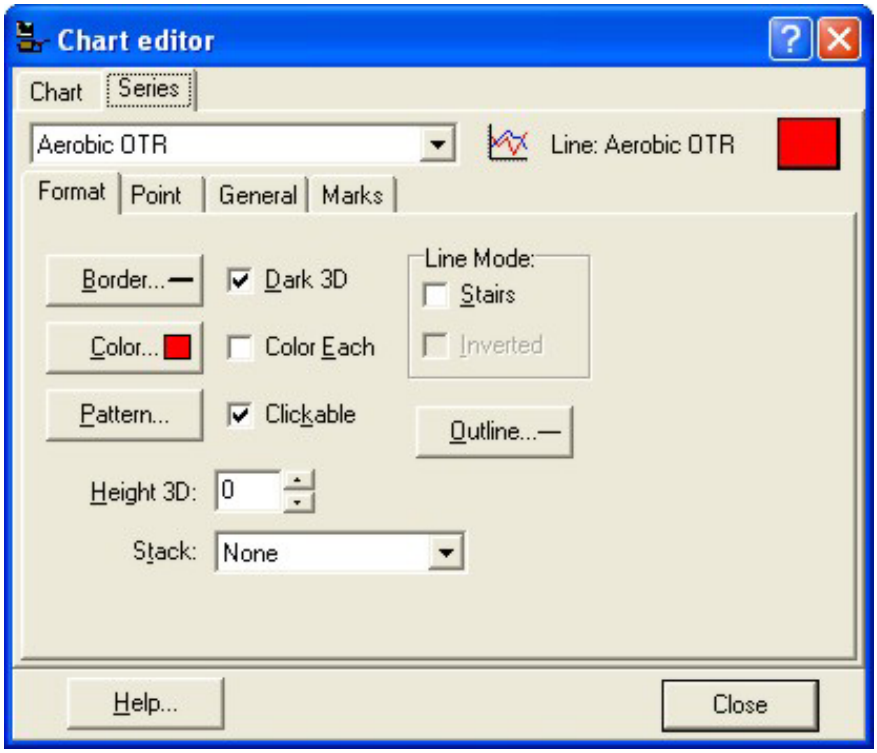


Line series line formatting options

4. Click the **Color...** button to open the color selection dialog box.
5. Click on the line series color you want and then click **OK** to close the color selection dialog box.
6. Click the **Close** button to finish.

Change the Line Series Border

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop-down list on the **Series** tab, select the line series you want to change the appearance of.
3. Click the **Format** sub-tab.

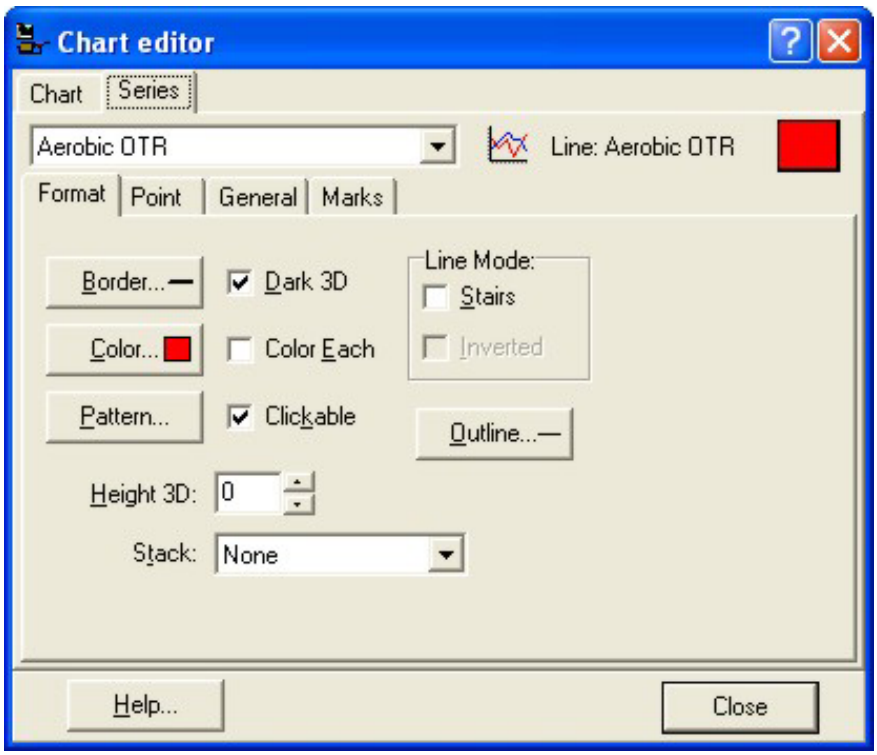


Line series line formatting options

4. Click the **Border...** button to open the border color editor.
5. Make sure that the box labeled **Visible** is checked.
6. Select the line series border style you want from the **Style** radio button group.
7. Select the line series border width you want from the **Width** spin edit box.
8. To change the line series border color, select the **Color...** button to open the color selection dialog box.
9. Click on the line series border color you want and then click **OK** to close the color selection dialog box.
10. Click **OK** to close the border color editor.
11. Click the **Close** button to finish.

Change the Line Series Pattern and Mode

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop-down list on the **Series** tab, select the line series you want to change the pattern or mode of.
3. Click the **Format** sub-tab.

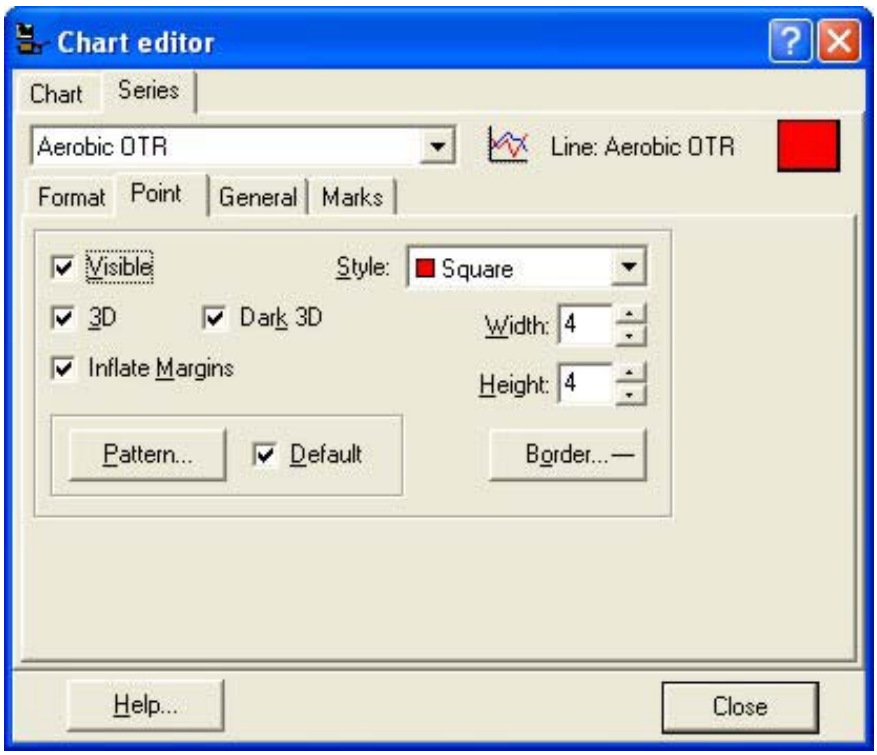


Line series line formatting options

4. To change the line series pattern, select a new one from the **Pattern…** button and click **OK** when finished.
5. If you want each instance of the chosen pattern to have a unique color, check the box labeled **Color Each**.
6. If you want the **Line Mode** to be such that the line is drawn in a stair-step manner between your data points, check the box labeled **Stairs**. To invert the stair-step pattern, check the box labeled **Inverted**.
7. Click the **Close** button to finish.

Change the Line Series Point General Appearance

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop-down list on the **Series** tab, select the line series you want to change point general appearance of.
3. Click the **Points** sub-tab.



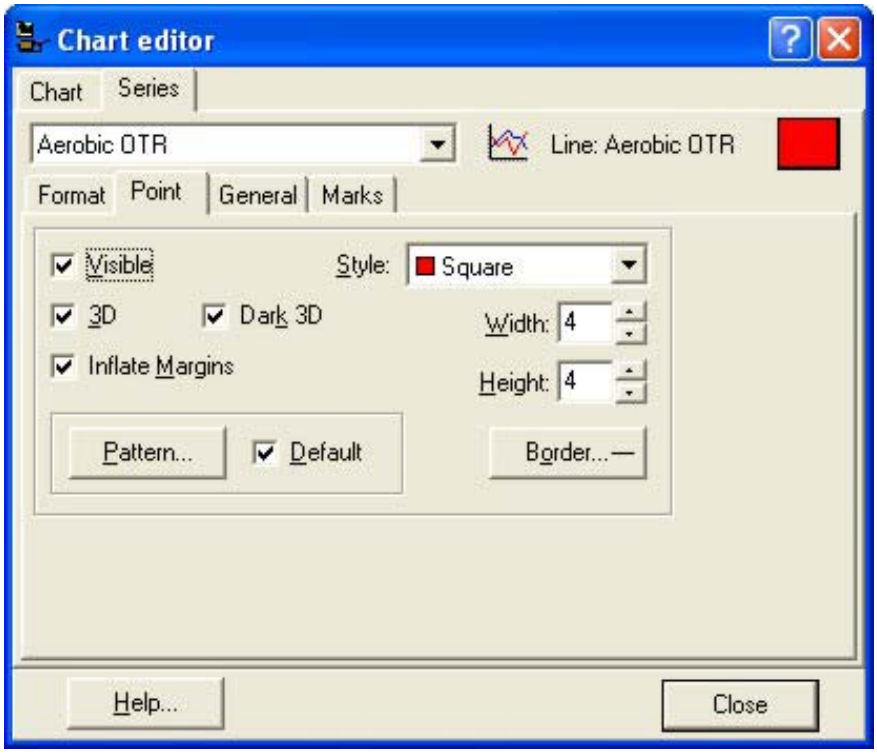
Line series point formatting options

- Ensure that the box labeled **Visible** is checked.
- Change the size of the points using the **Width** and **Height** spin edit boxes. To adjust the axes to accommodate the point size, check the box labeled **Inflate Margins**.
- To change the point marker, use the **Style:** drop list box to select a new one.
- If you choose **Square** point markers, you can give them a three-dimensional appearance by checking the box labeled **3D**.

7. Click the **Close** button to finish.

Change the Line Series Point Color

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop-down list on the **Series** tab, select the line series you want to change point general appearance of.
3. Click the **Points** sub-tab.

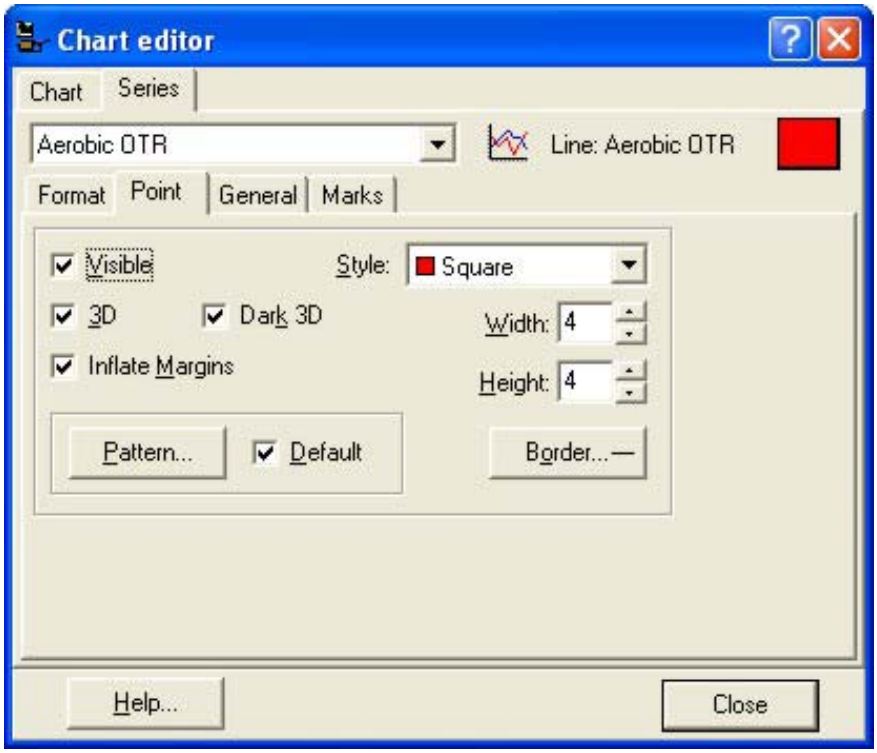


Line series point formatting options

4. Ensure that the box labeled **Visible** is checked.
5. Click the **Pattern...** button to open the pattern selection dialog box.
6. Click the **Color…** button to open the color selection dialog box.
7. Click on the line series point color you want and then click **OK** to close the color selection dialog box.
8. Click the **Close** button to finish.

Change the Line Series Point Border

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop-down list on the **Series** tab, select the line series you want to change point general appearance of.
3. Click the **Points** sub-tab.



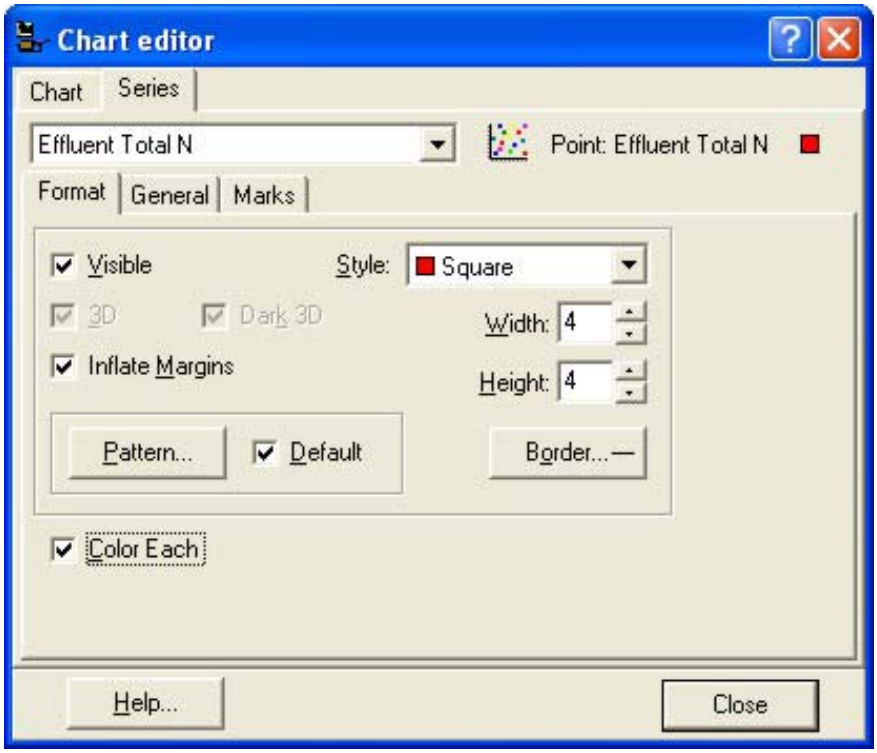
Line series point formatting options

4. Ensure that the box labeled **Visible** is checked.
5. Click the **Border...** button to open the border color editor.
6. Make sure that the box labeled **Visible** is checked.
7. Select the line series point border style you want from the **Style** radio button group.
8. Select the line series point border width you want from the **Width** spin edit box.
9. To change the line series point border color, select the **Color...** button to open the color selection dialog box.
10. Click on the line series point border color you want and then click **OK** to close the color selection dialog box.
11. Click **OK** to close the border color editor.
12. Click the **Close** button to finish.

Point Series Procedures

This is an overview of procedures that can be executed from the **Format** sub-tab of the **Edit Series** tab for a **Point Series**.

Point series are useful for X-Y plots, especially if the data are scattered or irregular. Quite often they are used to represent experimental data on a plot comparing observed and predicted results.



Point series formatting options

Change the Point Series General Appearance

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the point series you want to change the appearance of.
3. Click the **Format** sub-tab.
4. Ensure that the box labeled **Visible** is checked.
5. Change the size of the points using the **Width** and **Height** spin edit boxes. To adjust the axes to accommodate the point size, check the box labeled **Inflate Margins**.
6. To change the point marker, use the **Style:** drop list box to select a new one.
7. If you choose **Square** point markers, you can give them a three-dimensional appearance by checking the box labeled **3D**.
8. If you want each point in the series to have a random, different color, check the box labeled **Color Each**.
9. Click the **Close** button to finish.

Change the Point Series Pattern

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the point series you want to change the color of.

3. Click the **Format** sub-tab.
4. Ensure that the box labeled **Visible** is checked.
5. Click the **Pattern...** button to open the pattern selection dialog box.
6. Click on the pattern you want and then click **OK** to close the pattern selection dialog box.
7. Click the **Close** button to finish.

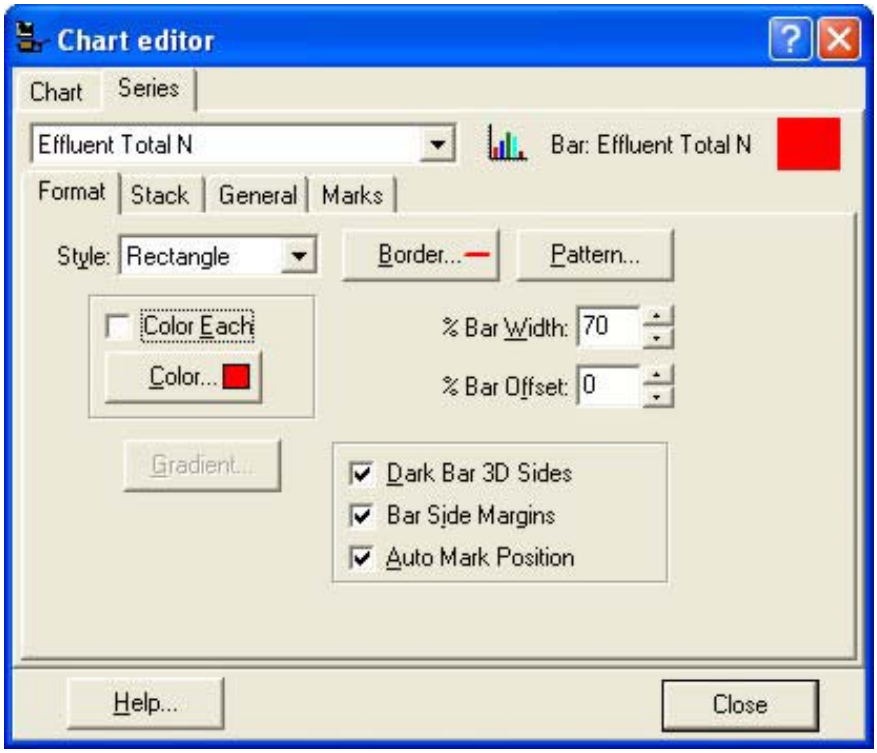
Change the Point Series Border

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the point series you want to change the border of.
3. Click the **Format** sub-tab.
4. Ensure that the box labeled **Visible** is checked.
5. Click the **Border...** button to open the border color editor.
6. Make sure that the box labeled **Visible** is checked.
7. Select the point border style you want from the **Style** radio button group.
8. Select the point border width you want from the **Width** spin edit box.
9. To change the point border color, select the **Color...** button to open the color selection dialog box.
10. Click on the point border color you want and then click **OK** to close the color selection dialog box.
11. Click **OK** to close the border color editor.
12. Click the **Close** button to finish.

Bar Series Procedures

This is an overview of formatting procedures that can be executed from the **Format** sub-tab of the **Edit Series** tab for a **Bar Series (Vertical and Horizontal)**. Note that when adding a new bar series to a chart, BioWin assumes that the bar series will consist of vertical bars. If this is not suitable for your chart, then you may change the bar series so that it uses horizontal bars by using the **Change…** option.

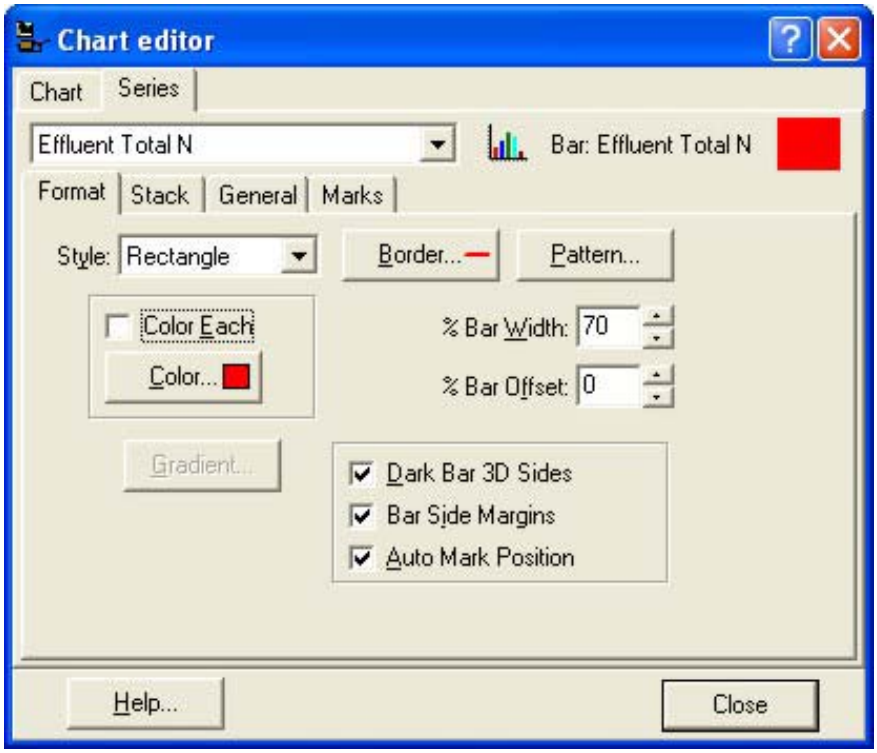
Bar series are useful for current value plots and displaying steady-state simulation results. Multiple bar series can be "stacked" to illustrate proportional differences.



Bar series formatting options

Change the Bar Series Color

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the bar series you want to change the color of.
3. Click the **Format** sub-tab.

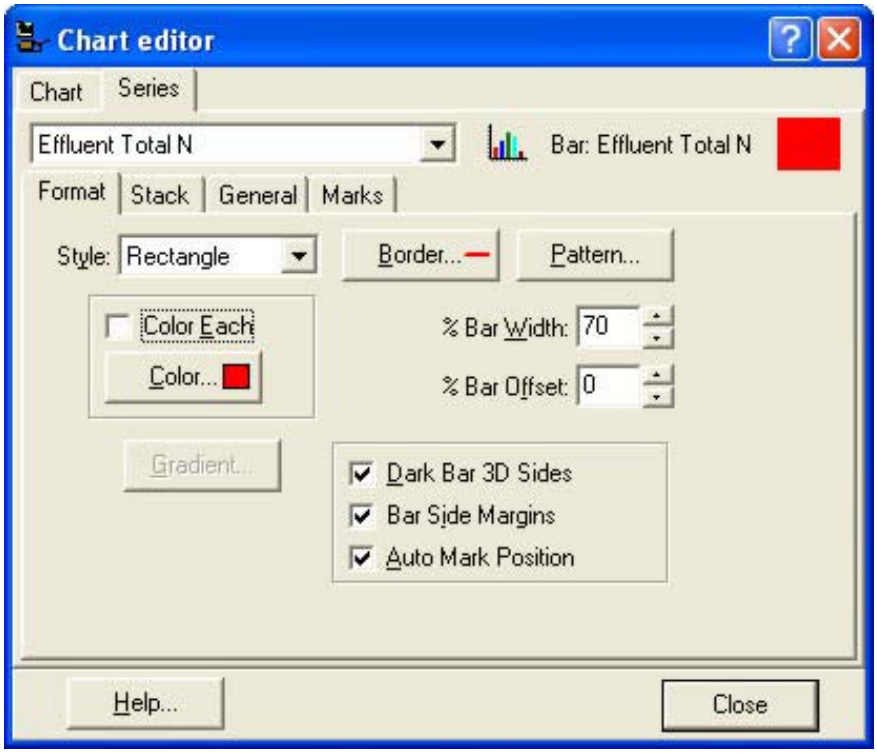


Bar series formatting options

4. If you want each bar in the series to have a random, different color, check the box labeled **Color Each**.
5. If you want all the bars to have the same color, click the **Color...** button to open the color selection dialog box.
6. Click on the bar color you want and then click **OK** to close the color selection dialog box.
7. If your bar series is three-dimensional and you want to have a shadow effect on the three-dimensional face, check the box labeled **Dark Bar 3D Sides**.
8. Click the **Close** button to finish.

Change the Bar Series Border

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the bar series you want to change the border of.
3. Click the **Format** sub-tab.

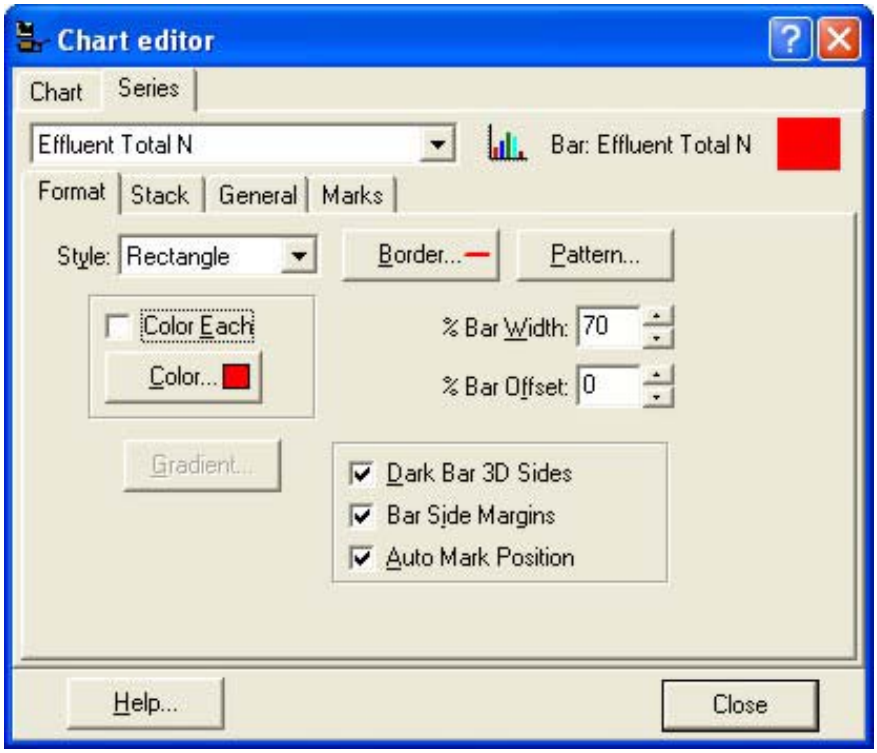


Bar series formatting options

4. Click the **Border...** button to open the border color editor.
5. Make sure that the box labeled **Visible** is checked.
6. Select the bar border style you want from the **Style** drop-down list box.
7. Select the bar border width you want from the **Width** spin edit box.
8. To change the bar border color, select the **Color...** button to open the color selection dialog box.
9. Click on the bar border color you want and then click **OK** to close the color selection dialog box.
10. Click **OK** to close the border color editor.
11. Click the **Close** button to finish.

Change the Bar Series Pattern

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the bar series you want to change the pattern of.
3. Click the **Format** sub-tab.

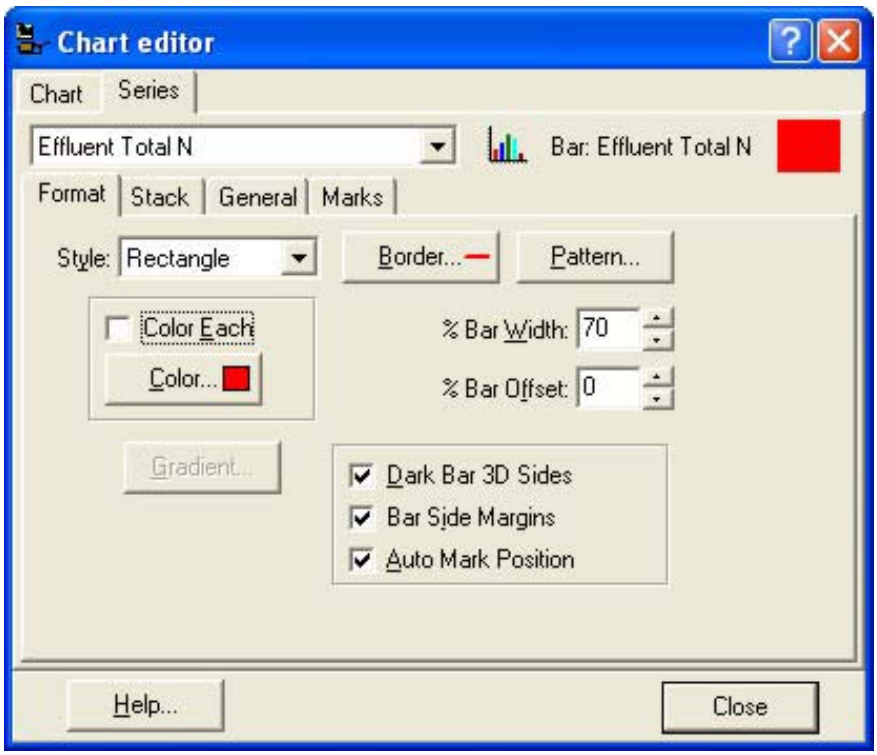


Bar series formatting options

4. Click the **Pattern...** button to open the fill pattern editor.
5. Make sure that the box labeled **No pattern** is un-checked.
6. Select the pattern style you want from the **Style** radio button group.
7. To change the pattern color, select the **Color...** button to open the color selection dialog box.
8. Click on the pattern color you want and then click **OK** to close the color selection dialog box.
9. Click **OK** to close the fill pattern editor.
10. Click the **Close** button to finish.

Change the Bar Series Shapes and Positions

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the bar series you want to change the shape and/or position of.
3. Click the **Format** sub-tab.



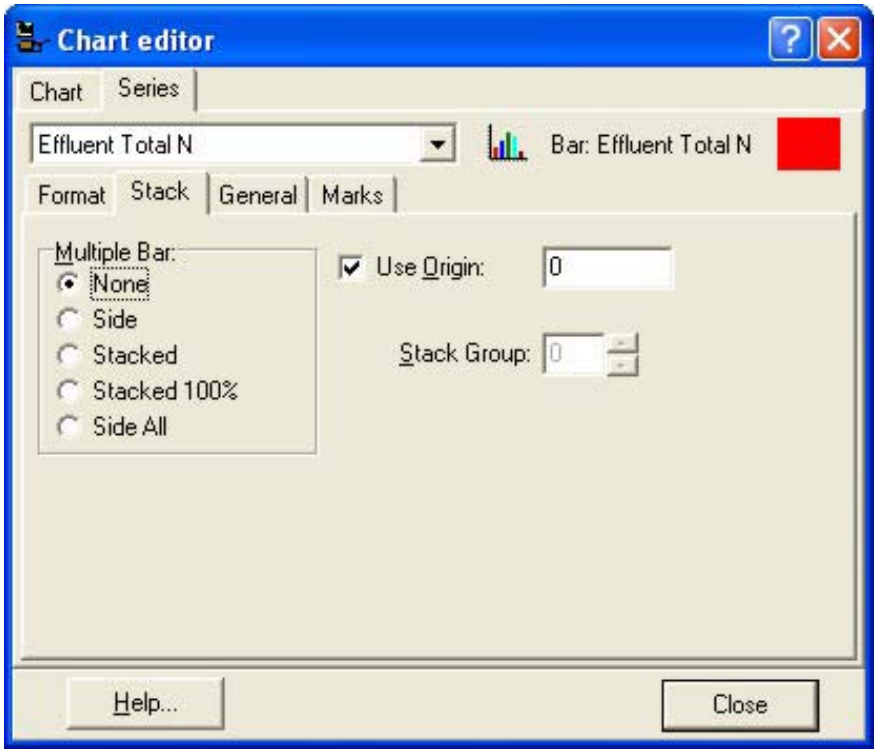
Bar series formatting options

- To change the shape of the bars in your bar series, use the **Style:** drop list box to choose from a variety of shapes.
- You can change the amount of space between the bars in a series by changing the value in the **% Bar Width:** spin edit box. Increasing the value increases the width of the bars, thus decreasing the space between them.
- You can shift the position of the bars in a series using the **% Bar Offset:** spin edit box. This property especially is useful when plotting multiple bar series on the same chart.
- If you want margins between the bars in a series and the chart boundaries, check the box labeled **Bar Side Margins**.
- If you are labeling the bars in a series you can reduce the amount that labels will overwrite each other by checking the box labeled **Auto Mark Position**.

4. Click the **Close** button to finish.

Control Bar Placement for Multiple Bar Series

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select one of the bar series. In this special case of series formatting, selecting one bar series also will result in changes to the other bar series.
3. Click the **Stack** sub-tab.



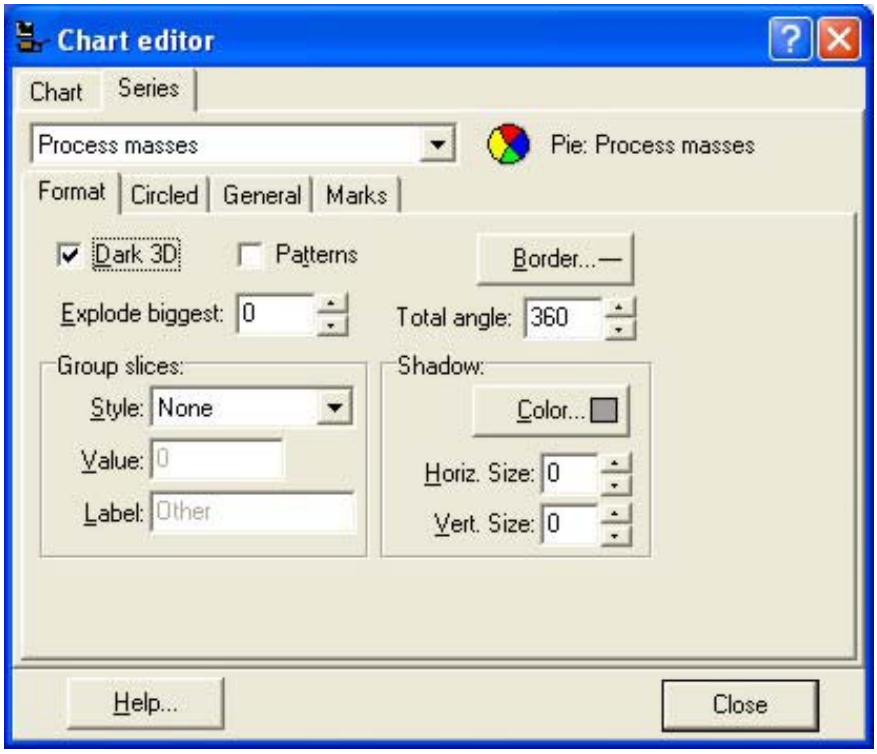
Bar series stack options

4. In the **Multiple Bar:** radio button group choose one of the following options for bar placement:
   - **None:** This will place corresponding bars in each series in front of one another. However, if your chart is three-dimensional you will be able to see portions of all the series.
   - **Side:** This will place corresponding bars in each series beside one another.
   - **Stacked:** This will place corresponding bars in each series on top of one another.
   - **Stacked 100%:** This will place corresponding bars in each series on top of one another showing the relative size of each bar on a 100 % scale.
   - **Side All:** This will place each complete series side by side.
5. To have bar bottoms for a series start at the zero coordinate, check the box labeled Use Origin. To have them begin at a different ordinate value, uncheck this box and input the desired starting value.
6. Click the **Close** button to finish.

Pie Series Procedures

This is an overview of formatting procedures that can be executed from the **Format** sub-tab of the **Edit Series** tab for a **Pie Series**.

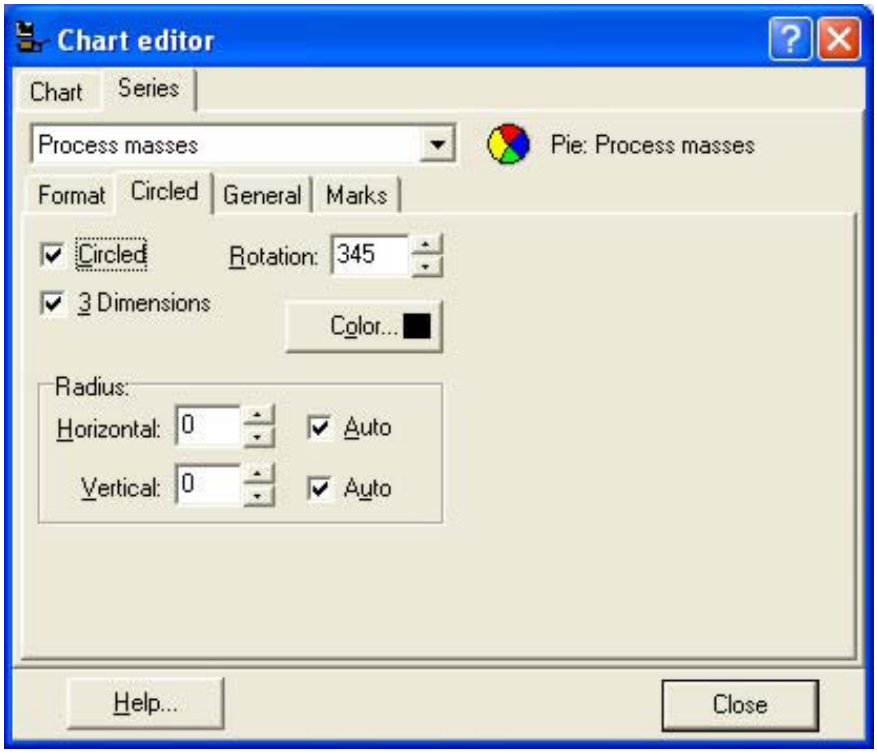
Pie series are useful for displaying proportional or fractional data when you desire to show the size of the portions relative to each other. For example, pie series may be used to display wastewater characteristic fractions, plant sludge mass fractions, plant volume fractions, etc. They also are a good method for displaying results in current value plots.



Pie series formatting options

Change the Pie Series General Appearance

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the pie series you want to change the appearance of.
3. Click the **Circled** sub-tab.



Pie series circled options

- If you want your pie series to be circular in shape, check the box labeled **Circled Pie**.
- If you want to have an elliptical pie series, adjust the dimensions using the **Horizontal** and **Vertical** spin edit boxes in the **Radius** section. To let BioWin determine your elliptical radii, check the boxes labeled **Auto**.
- To rotate the pie chart, change the value in the **Rotation:** spin edit box.

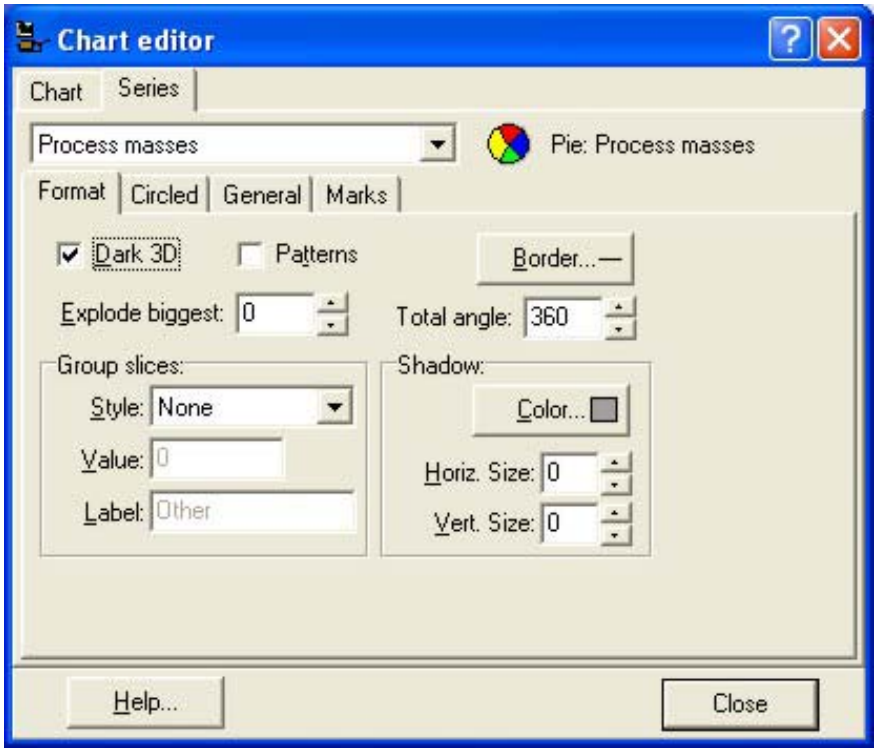
4. Click the **Close** button to finish.

Change the Pie Series Color and Pattern

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the pie series you want to change the appearance of.
3. Click the **Format** sub-tab.
4. If you would rather have patterns as opposed to solid color fills for your pie slices, check the box labeled **Patterns**.
5. If you choose to have pattern fills, you can change the underlying pie color. Click the **Back Color...** button to open the color selection dialog box, select the pie color you want, and click **OK**.
6. Click the Close button to finish.

Change the Pie Series Border

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the pie series you want to change the border of.
3. Click the **Format** sub-tab.



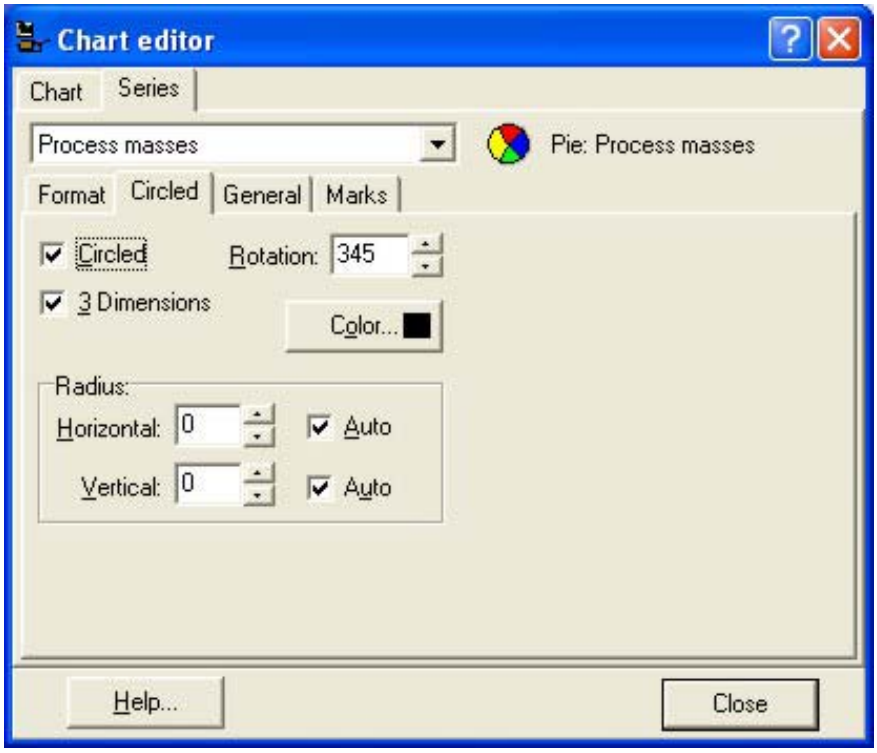
Pie series formatting options

4. Click the **Border...** button to open the border color editor.
5. Make sure that the box labeled **Visible** is checked.
6. Select the pie series border style you want from the **Style** drop-down list box.
7. Select the pie series border width you want from the **Width** spin edit box.
8. To change the pie series border color, select the **Color...** button to open the color selection dialog box.
9. Click on the pie series border color you want and then click **OK** to close the color selection dialog box.
10. Click **OK** to close the border color editor.
11. Click the **Close** button to finish.

Change the Pie Series Three-Dimensional Appearance

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.

2. In the **Series** drop list box, select the pie series you want to change the three-dimensional appearance of.
3. Click the **Circled** sub-tab.



Pie series circled options

4. If you want your pie series to be three-dimensional, check the box labeled **3 Dimensions**.
5. If you want the three-dimensional pie face to have color, check the box labeled **Color**. To give this color a slightly darker tint than the top of the pie series, check the box labeled **Dark 3D** on the **Format** tab.
6. To have a darker shadow effect on the three-dimensional pie face, increase the value in the **Horiz. Size** and **Vert. Size** spin edit boxes in the section labeled **Shadow**.
7. To change the shadow color, click the **Color...** button to open the color selection dialog box, select the shadow color you want, and click **OK**.
8. Click the **Close** button to finish.

Arrange the Pie Series Slices

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the pie series you want to change the three-dimensional appearance of.
3. Click the **Format** sub-tab.

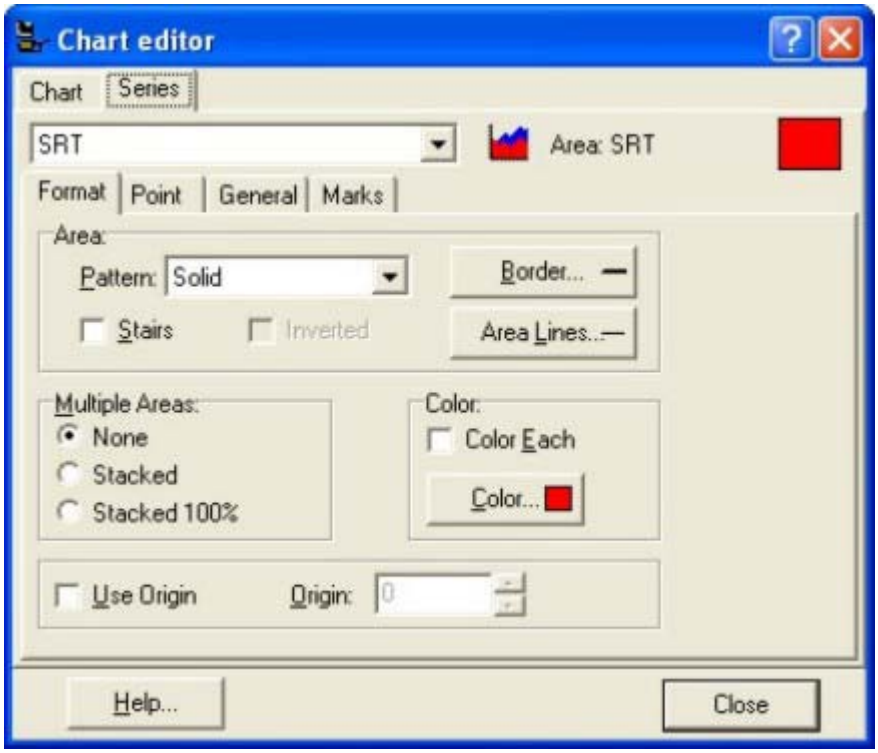
- If you want to place emphasis on the largest pie slice, use the **Explode biggest** spin edit box to separate the largest slice from the rest of the pie.
- You can change the arrangement order of the pie slices using the **Order** drop list box.
- You can combine all pie slices below a specified value or percentage into a single slice. To do this:

1. Use the **Style** drop list box to choose the criteria (below a certain value or percent) you want to use to group slices.
2. In the **Value** text edit area, enter the cut-off value. All pie slices below this value will be combined into one pie.
3. You can label the grouped slices. Click the **Labels** tab and ensure that **Show Series Labels** is checked. Now click back on the **Format** tab and enter your text in the **Label** text edit area at the bottom of the dialog box.

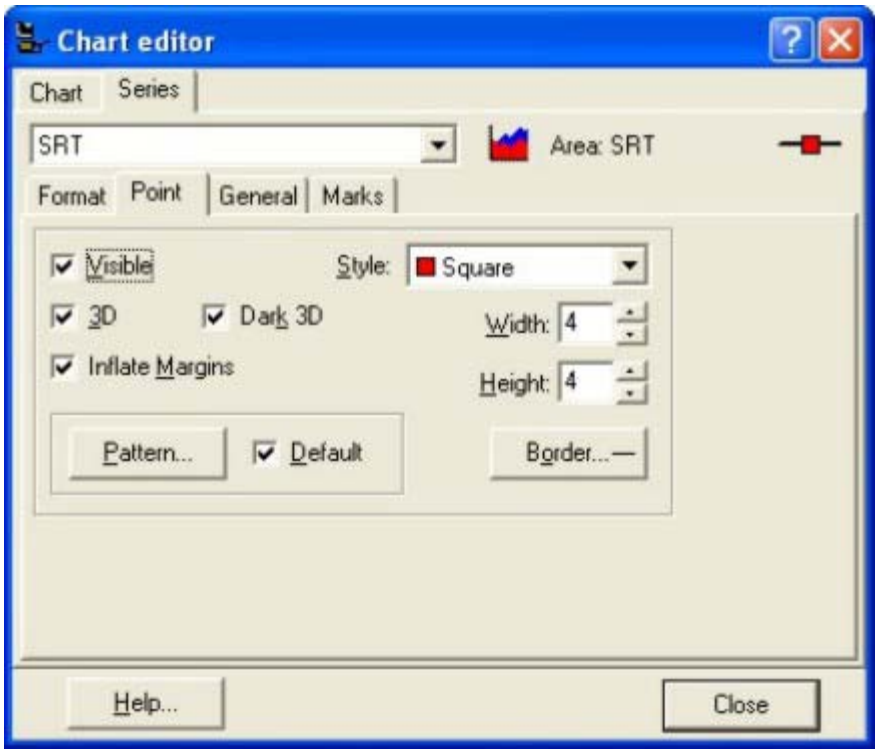
Area Series Procedures

This is an overview of formatting procedures that can be executed from the **Format** and **Points** sub-tabs of the **Edit Series** tab for an **Area Series**.

Area series are somewhat of a combination of a line chart and a bar chart. They provide a richer way of displaying line chart data in three dimensions, and offer some of the stacking properties of multiple bar series to illustrate proportional differences. As such they are useful for displaying results in current value charts.



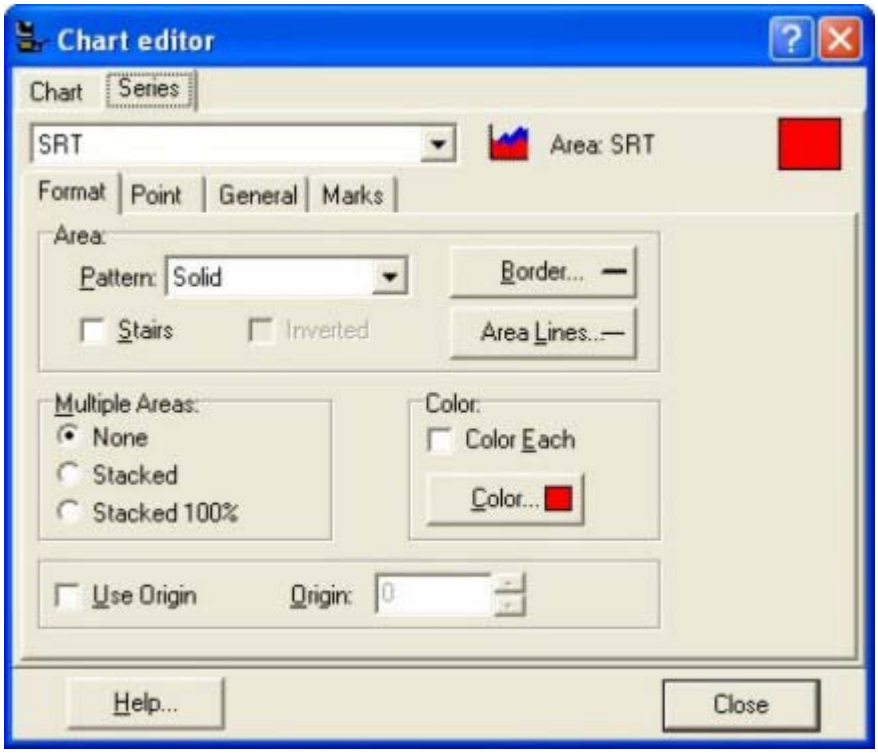
Area series formatting options



Area series point formatting options

Change the Area Series General Appearance

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the area series you want to change the appearance of.
3. Click the **Format** sub-tab.



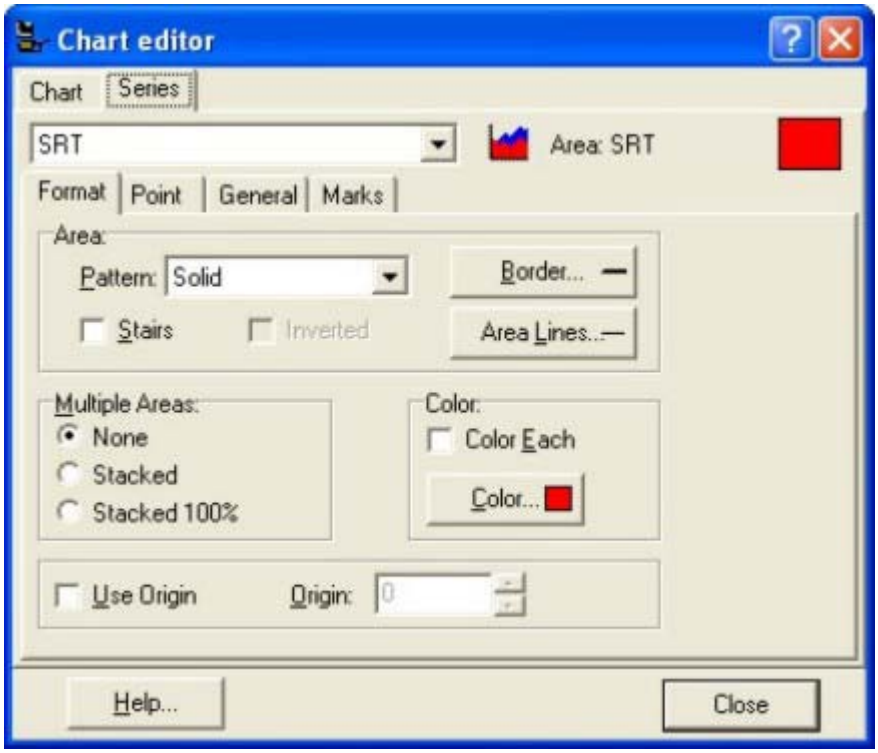
Area series format options

- If you want a pattern (drawn in black lines) on the face of your area series, use the **Pattern** drop list box to select the one you desire.
- To have the points that define the area series joined in a "stair-step" pattern, check the box labeled **Stairs**.

4. Click the **Close** button to finish.

Change the Area Series Color

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the area series you want to change the color of.
3. Click the **Format** sub-tab.

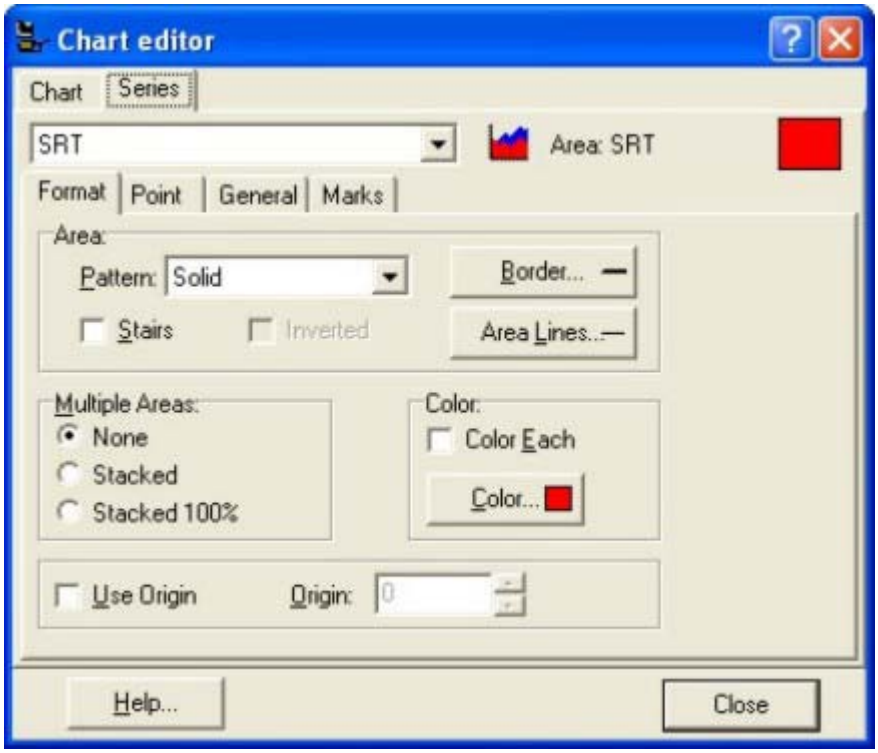


Area series format options

4. If you want each area segment in the series to have a random, different color, check the box labeled **Color Each**.
5. If you want all the area segments to have the same color, click the **Color...** button to open the color selection dialog box.
6. Click on the area color you want and then click **OK** to close the color selection dialog box.
7. Click the **Close** button to finish.

Change the Area Series Border

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the area series you want to change the border of.
3. Click the **Format** sub-tab.

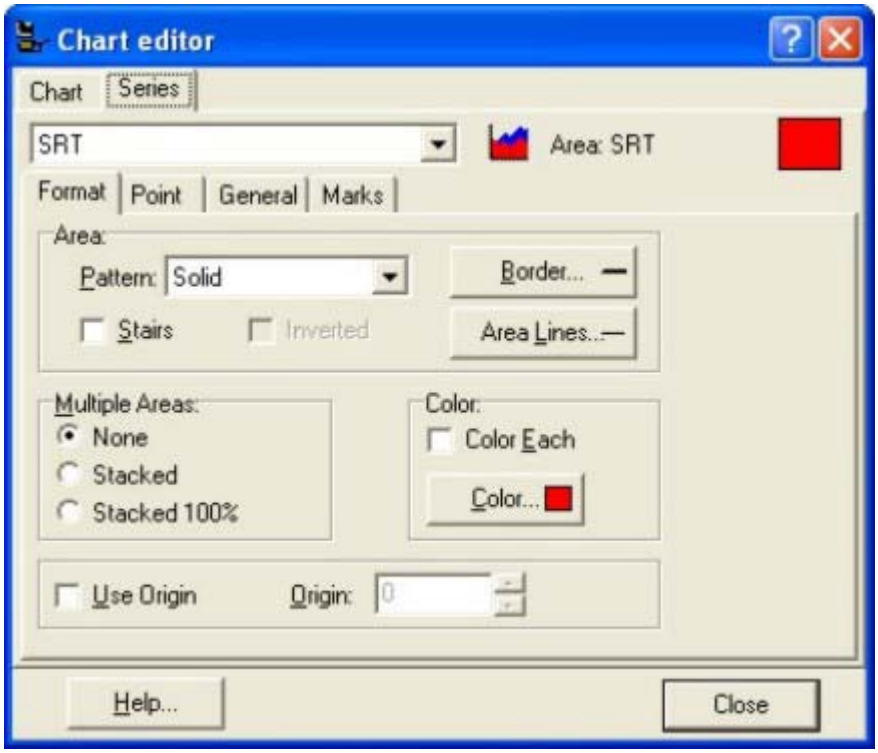


Area series format options

4. Click the **Border...** button to open the border color editor.
5. Make sure that the box labeled **Visible** is checked.
6. Select the area border style you want from the **Style** radio button group.
7. Select the area border width you want from the **Width** spin edit box.
8. To change the area border color, select the **Color...** button to open the color selection dialog box.
9. Click on the area border color you want and then click **OK** to close the color selection dialog box.
10. Click **OK** to close the border color editor.
11. Click the **Close** button to finish.

Change the Area Series Lines

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the area series you want to change the lines of.
3. Click the **Format** sub-tab.

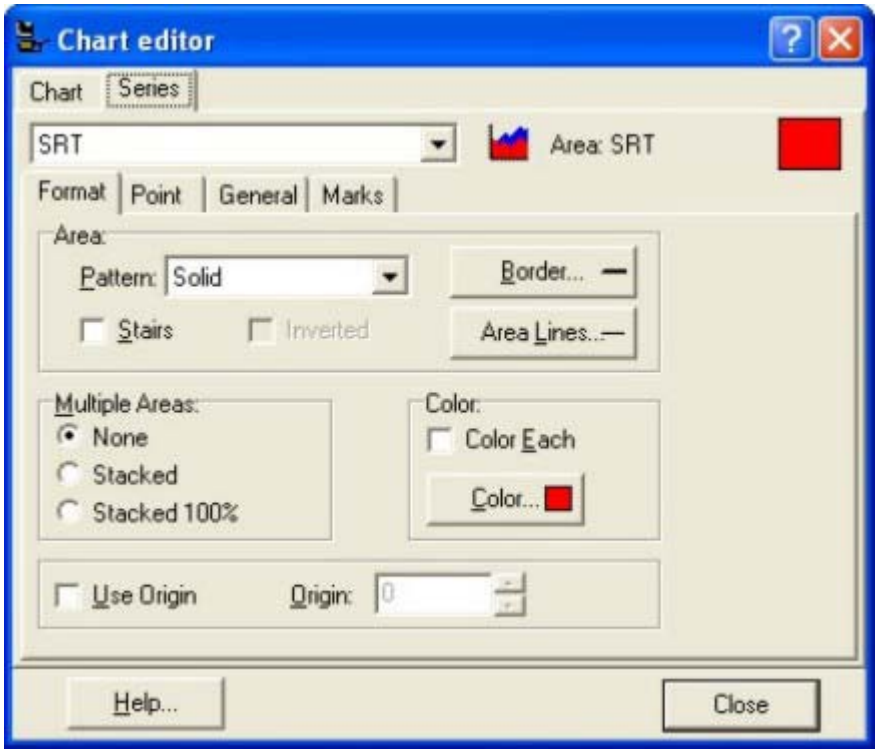


Area series format options

4. Click the **Area Lines...** button to open the border color editor.
5. Make sure that the box labeled **Visible** is checked.
6. Select the area line style you want from the **Style** radio button group.
7. Select the area line width you want from the **Width** spin edit box.
8. To change the area line color, select the **Color...** button to open the color selection dialog box.
9. Click on the area line color you want and then click **OK** to close the color selection dialog box.
10. Click **OK** to close the border color editor.
11. Click the **Close** button to finish.

Control Area Placement for Multiple Area Series

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select one of the area series. In this special case of series formatting, selecting one area series also will result in changes to the other area series.
3. Click the **Format** sub-tab.

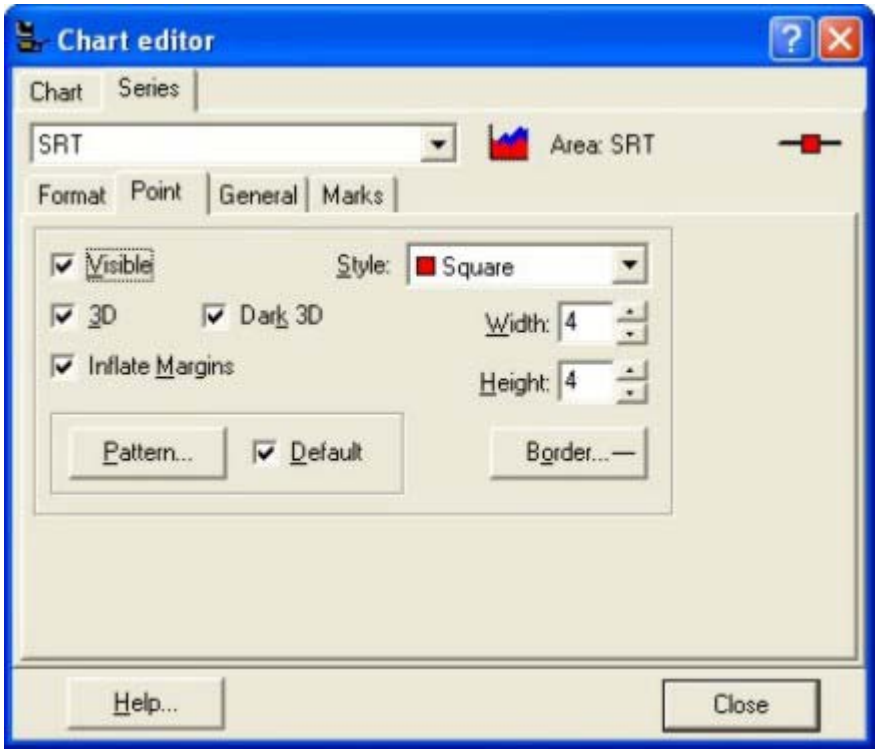


Area series format options

4. In the **Multiple Areas:** radio button group choose one of the following options for area series placement:
   - **None:** This will place each area series in front of one another. However, if your chart is three-dimensional you will be able to see portions of all the series.
   - **Stacked:** This will place each area series on top of one another.
   - **Stacked 100%:** This will place each area series on top of one another showing the relative size of each area on a 100 % scale.
5. Click the **Close** button to finish.

Change the Area Series Point General Appearance

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the area series you want to change the point general appearance of.
3. Click the **Point** sub-tab.



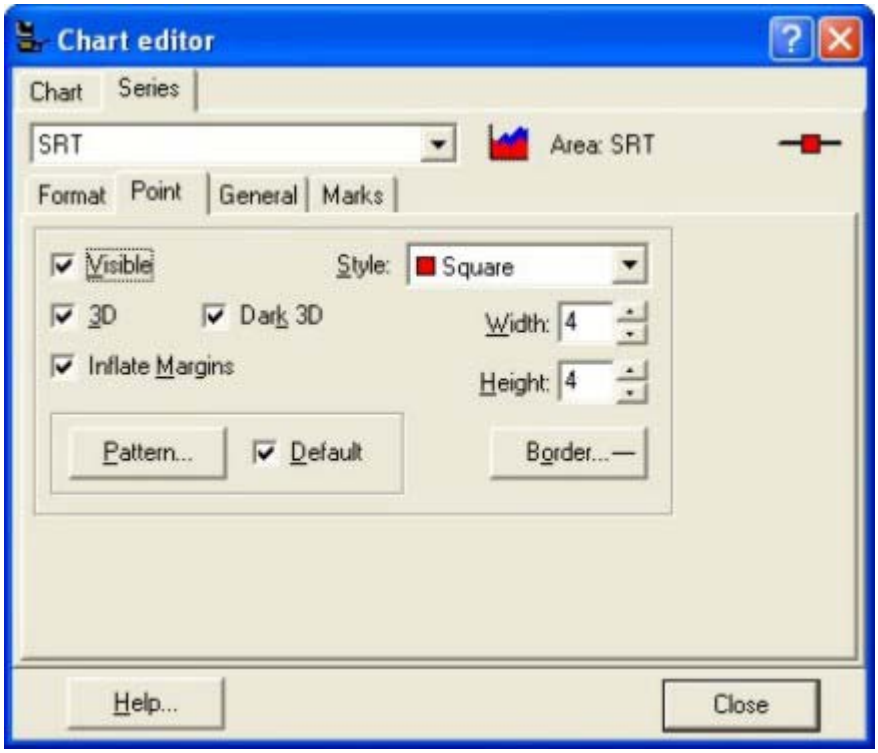
Area series point options

- Ensure that the box labeled **Visible** is checked.
- Change the size of the points using the **Width** and **Height** spin edit boxes. To adjust the axes to accommodate the point size, check the box labeled **Inflate Margins**.
- To change the point marker, use the **Style:** drop list box to select a new one.
- If you choose **Square** point markers, you can give them a three-dimensional appearance by checking the box labeled **3D**.
- To give this color a slightly darker tint than the top of the pie series, check the box labeled **Dark 3D.**

4. Click the **Close** button to finish.

Change the Area Series Point Pattern

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the area series you want to change the point color of.
3. Click the **Point** sub-tab.

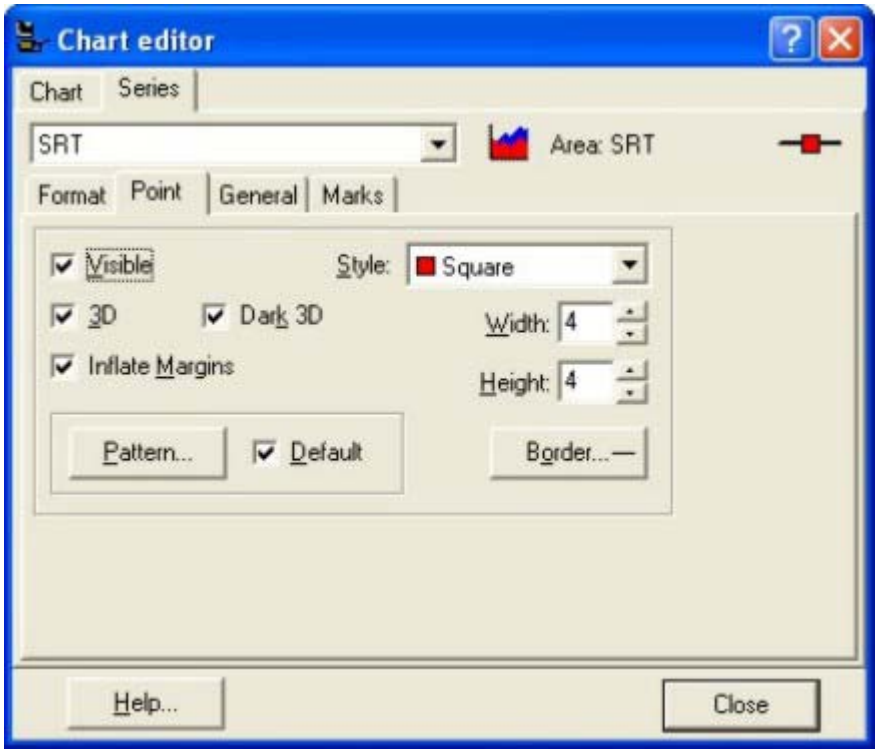


Area series point options

4. Ensure that the box labeled **Visible** is checked.
5. Click the **Pattern...** button to open the pattern selection dialog box.
6. Click on the area series point pattern you want and then click **OK** to close the color selection dialog box.
7. Click the **Close** button to finish.

Change the Area Series Point Border

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the area series you want to change the point border of.
3. Click the **Point** sub-tab.



Area series point options

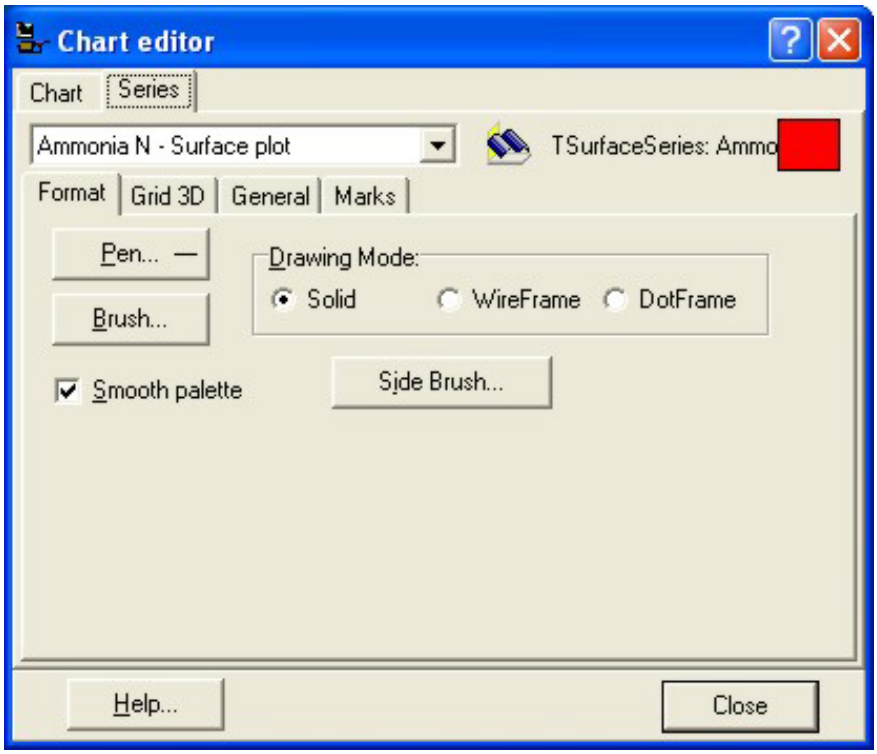
4. Ensure that the box labeled **Visible** is checked.
5. Click the **Border...** button to open the border color editor.
6. Make sure that the box labeled **Visible** is checked.
7. Select the area series point border style you want from the **Style** radio button group.
8. Select the area series point border width you want from the **Width** spin edit box.
9. To change the area series point border color, select the **Color...** button to open the color selection dialog box.
10. Click on the area series point border color you want and then click **OK** to close the color selection dialog box.
11. Click **OK** to close the border color editor.
12. Click the **Close** button to finish.

Surface Series Procedures

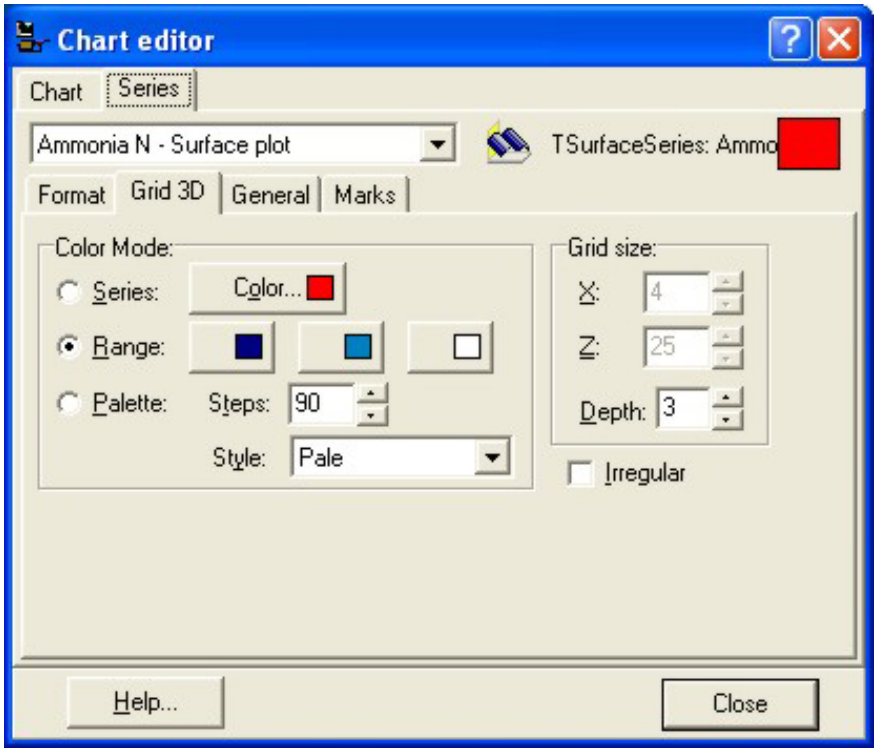
This is an overview of formatting procedures that can be executed from the **Format** and **Grid** sub-tabs of the **Edit Series** tab for a **Surface Series**.

Surface series can be used for spatial profiles of a given compound. For example, say you have a configuration with three bioreactors, and you want to show the ammonia profile over the bioreactors. The ammonia concentration would be plotted on the Y (vertical) axis, the bioreactor number would be plotted on the X (horizontal)

axis, and time would be plotted on the Z axis. They also are useful for displaying time-histories of secondary settler depth - concentration profiles.



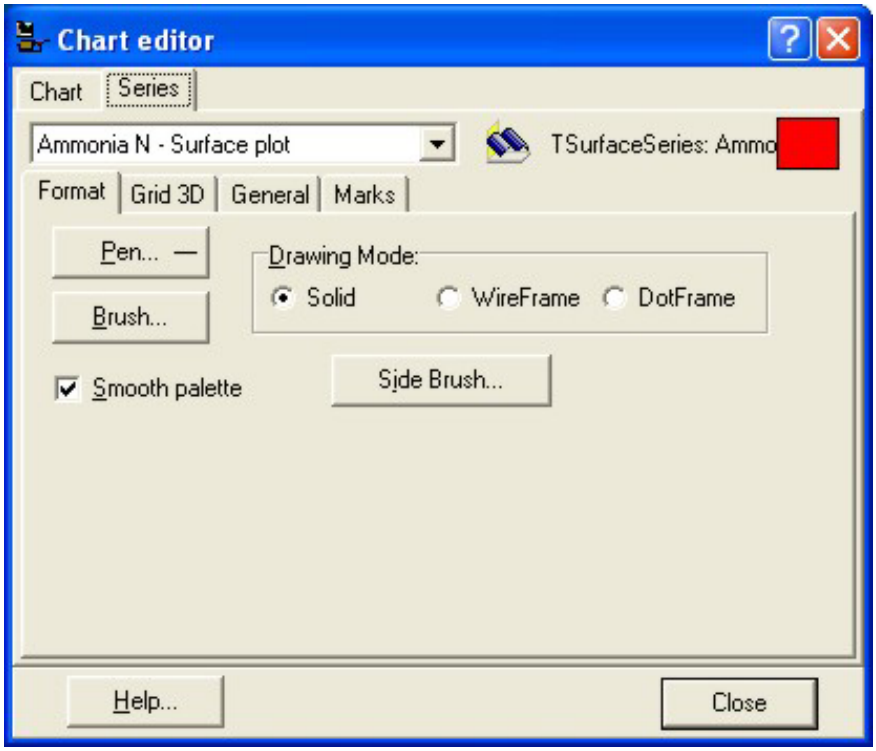
Surface series formatting options



Surface series grid options

Surface Series Gridline Appearance

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the surface series you want to change the gridlines of.
3. Click the **Format** sub-tab.

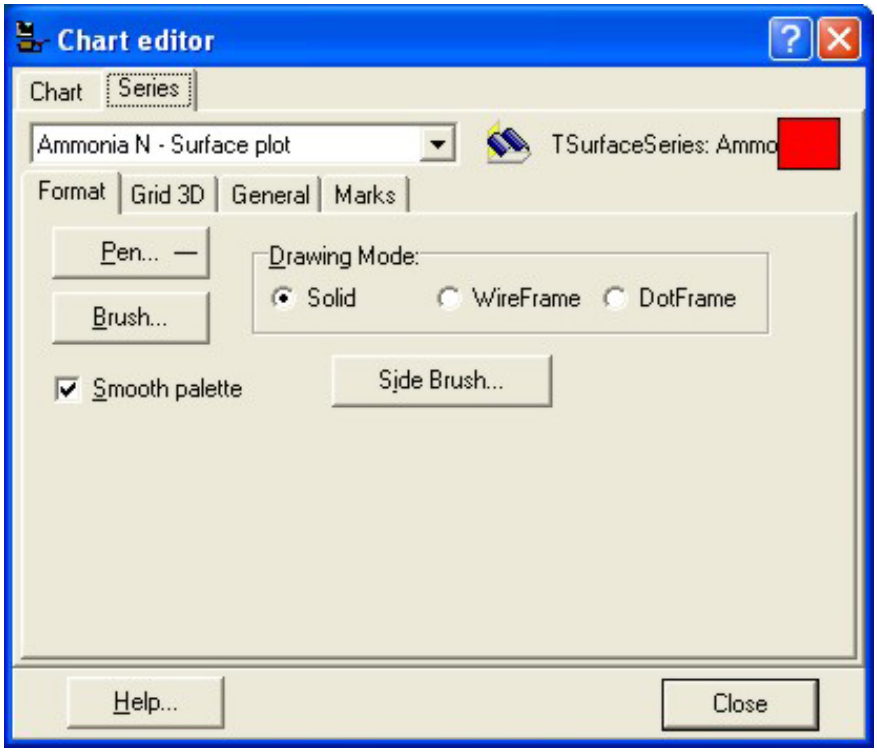


Surface series formatting options

4. Click the **Pen…** button to open the border color editor.
   - To turn the gridlines on or off, check or uncheck the box labeled **Visible**.
   - To change the gridline thickness, increase or decrease the value in the **Width** spin edit box.
   - To change the line style (e.g. solid, dashed, etc.) used to draw the gridlines, click the appropriate radio button in the **Style** radio button group. Note that you may not have a **Width** greater than 1 with any style other than **Solid**.
   - To change the gridline color button, click the **Color** button to open the color selection dialog box.
5. When you are satisfied with the appearance of your surface series gridlines, click **OK** to close the border color editor.
6. Click the **Close** button to finish.

Surface Series Drawing Mode

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the surface series you want to change the drawing mode of.
3. Click the **Format** sub-tab.



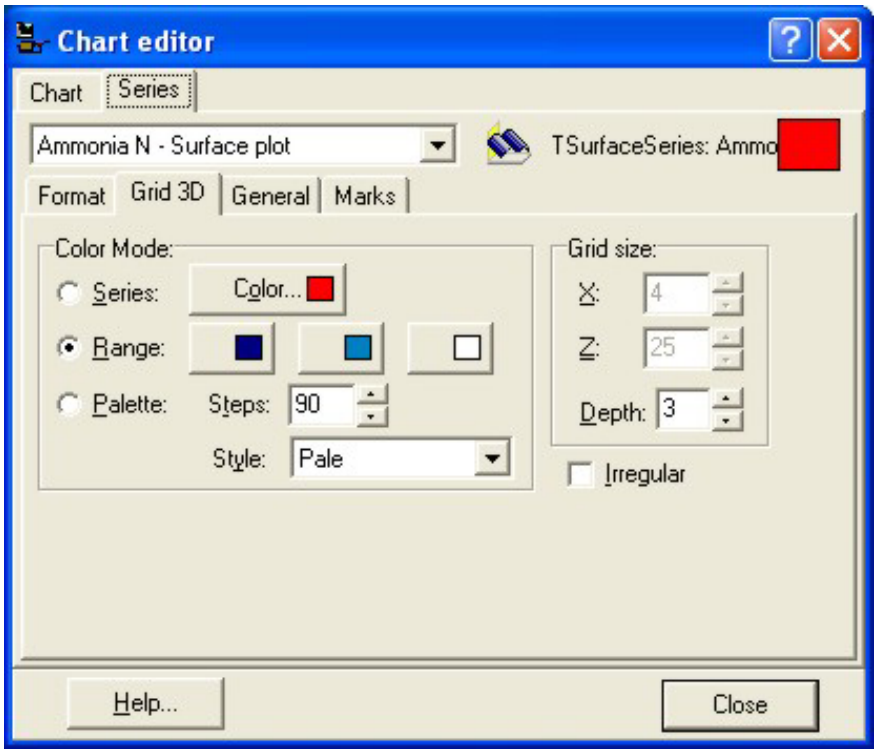
Surface series formatting options

4. From the **Drawing Mode** radio button group, choose from the following:
   - To have a colored surface, choose **Solid** mode.
   - To have a surface with only the gridlines showing, choose **WireFrame** mode.
   - To have your surface drawn with a series of small dots, choose **DotFrame** mode. Note that this mode may be difficult to see with some color combinations.
5. When you are satisfied with the appearance of your surface series, click the **Close** button to finish.

Surface Series Color Mode

1. Right-click on the chart to open the **Edit** menu, and then click **Edit Series**.
2. In the **Series** drop list box, select the surface series you want to change the color mode of.

3. Click the **Grid 3D** sub-tab.



Surface series grid options

4. From the **Color Mode** radio button group, choose from the following:
   - To have one color for the entire surface, choose **Series Color** mode. Click on the color button next to this option to open the color selection dialog box if you want to change the color.
   - To have a gradient color fill for the surface, choose **Range** mode. Click on the **From:** and **To:** color boxes to change the colors used. The **From:** color is assigned to points with the largest Y value (i.e. the "high" points in the surface), and the **To:** color is assigned to points with the lowest Y value (i.e. the "low" points in the surface).
   - To have a stepped color fill for the surface, choose **Palette** mode. The value in the **Steps** spin edit box determines the range of colors used in the surface fill, and the height of each point in the surface determines its assigned color from that range.
5. When you are satisfied with the appearance of your surface, click **Close** to finish.

Managing BioWin Projects

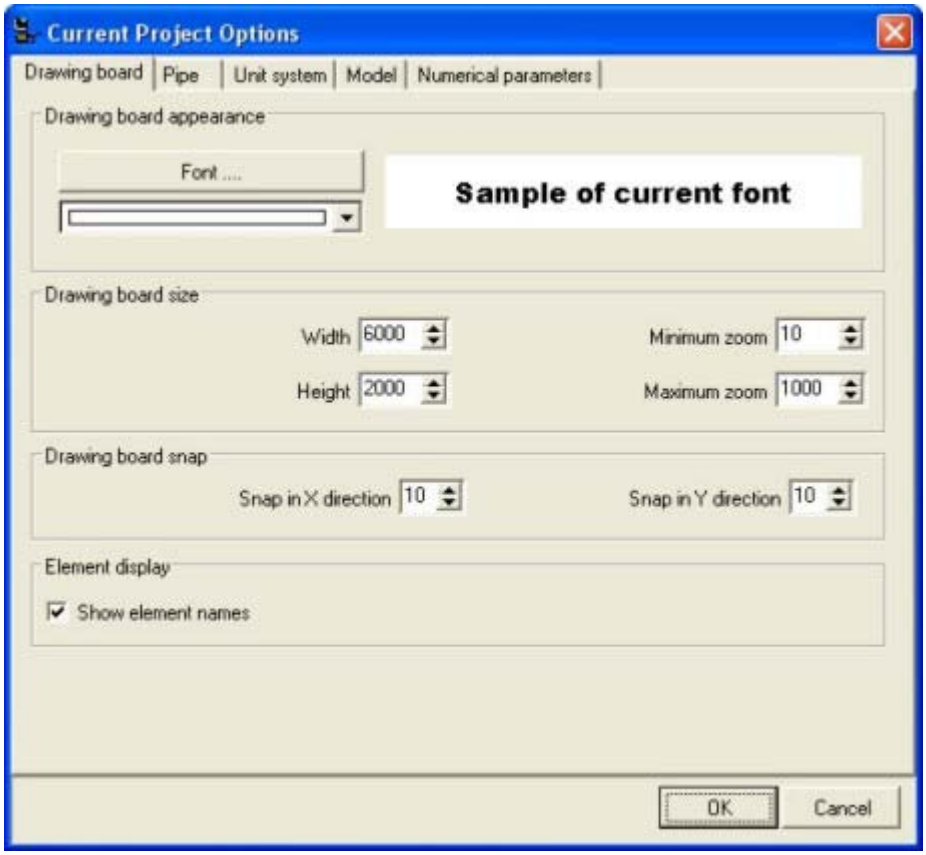
Setting Project Options

Project options are specific to the current project. When you set current project options, they will override any similar settings that you have applied to your BioWin defaults with the **Project|New Project Options** command. For example, if you have set your default flowsheet color to be blue with the **Project|New Project Options** command, but for "Project A" you want the drawing board color to be white, then you would set the drawing board color to white in "Project A" using the **Project|Current Project Options** command. Now the drawing board color for "Project A" will be white, regardless of the default values you have set.

Since project options are file specific, they "travel" with that file. For example, if you define a set of project options for "Project A" on your copy of BioWin and then open the "Project A" file in a colleague's copy of BioWin, you still will see your defined project options. As before, these project options will override any similar settings that the owner of the other copy of BioWin has set as defaults using the **Project|New Project Options** command.

Drawing Board Options

This section outlines the various project options that can be set for the drawing board. The drawing board options may be accessed in the main simulator window by choosing **Project|Current Project Options** from the menu and selecting the **Drawing board** tab, shown in the picture below.



Current project drawing board options tab

Aspects of the **Drawing board appearance** that may be changed include the **Font** and the **Color**. Clicking the **Font…** button will open the font properties dialog box and will allow you to change the font that is used to label element pictures on the drawing board. Selecting a new color from the drop list box will change the background color of the drawing board. Notice that a preview of the drawing board background color and the selected font is given in the dialog box.

The **Drawing board size** also may be changed. Changing the values in the **Width** and **Height** spin edit boxes will change the overall dimensions of the drawing board. The size of the main window occupied by the drawing board is not changed, however, the overall size is changed as evidenced by a change in the size of the scroll box in the scroll bar. The default values of 6,000 by 2,000 translate roughly to 60 by 20 inches. It should be mentioned here that if you plan to be performing copy and paste actions with Microsoft Word, the size of drawing board that you can copy into Word is limited to 22 by 22 inches. The **zoom limits** of the drawing board can be set so that zooming can only take place between minimum and maximum levels.

The **Drawing board snap** resolution can be changed in either or both the X and Y directions. The snap feature helps you align elements precisely on the drawing board. When you place or move an element on the drawing board, it aligns itself (i.e. "snaps") to the nearest grid point (grid points are invisible). Therefore, increasing the snap values results in a coarser grid for the elements to snap to which means that you have less control over their placement. Decreasing the snap values results in a finer grid for the elements to snap to which means that you have increased control over their placement.

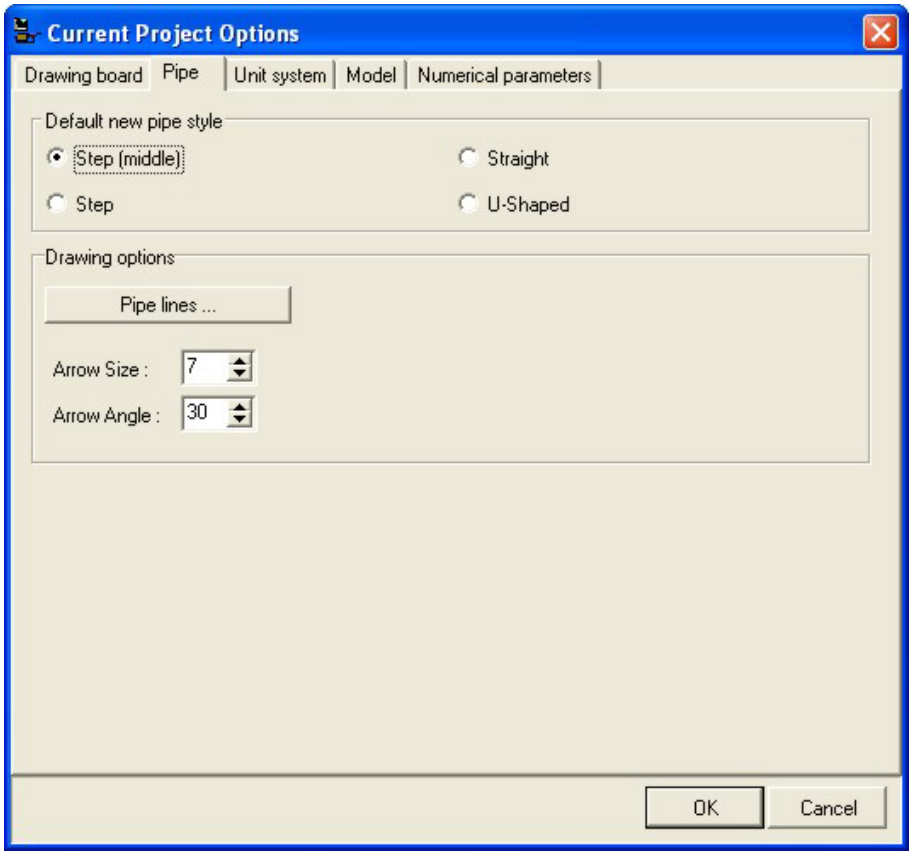
You can change the drawing board **Element display**. You can choose to have the display of **element names** either on or off.

Pipe Options

You can change the default style for new pipes in a project by choosing one of the four available styles in the **Default new pipe style** group. For information on the various pipe styles, see the **Pipes** topic in the *Element Types* chapter.

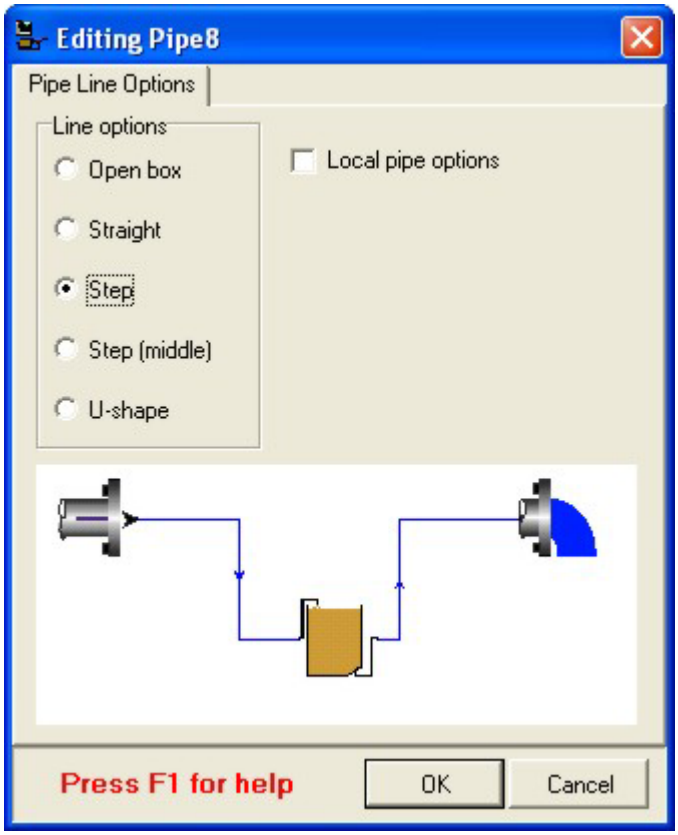
The "arrow angle" refers to the acute angle between the arrow side and the line. For example, if you want arrows with "flat" bases, set the arrow side angle to the maximum of 60 degrees.

You also can control the appearance of pipes in your project, using the **Project|Current Project Options** menu command and selecting the **Pipe** tab, shown in the figure below. To change the **Color**, **Width**, and **Style** of the lines used to represent pipes on the drawing board, click on the **Pipe Lines…** button. You may increase the **arrow size** on the lines used to represent pipes. The **arrow angle** also can be changed.



Current project pipe options tab

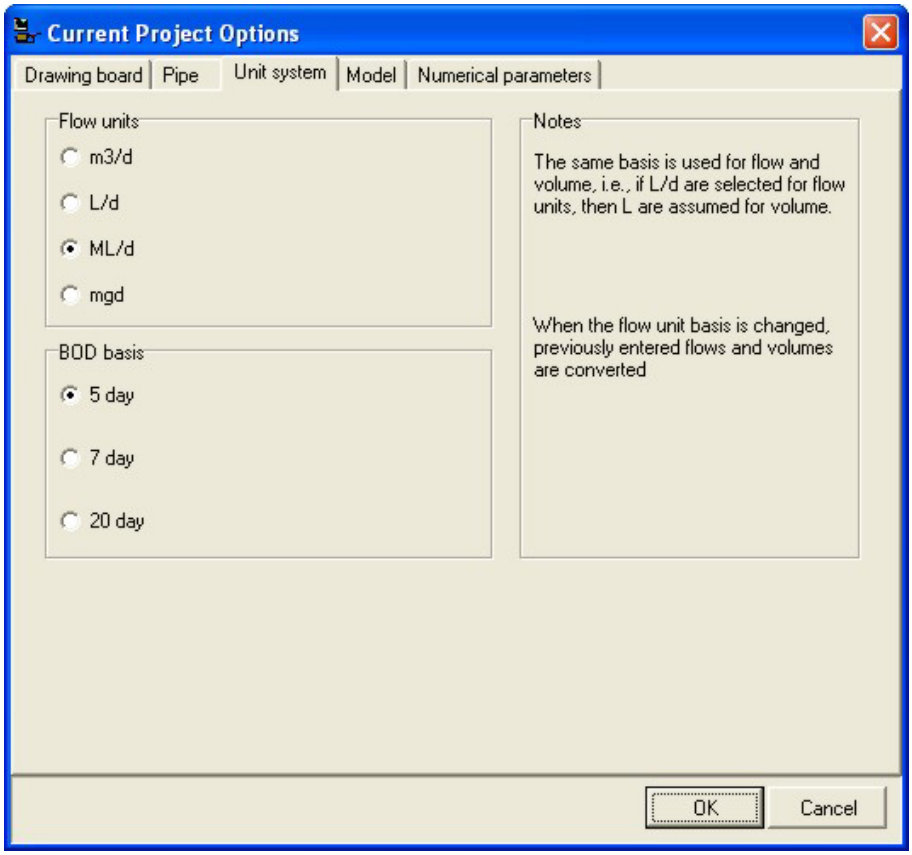
Note that it also is possible to change pipe line and arrow settings for individual pipes. Access the properties of the pipe you wish to change by double-clicking it or right-clicking it and selecting **Properties…** from the resulting pop-up menu. Next, check the box labeled **Local pipe options** and you will be presented with the same set of options that have just been outlined; however, these changes will affect this particular pipe only.



Pipe line options dialog box

Unit System

This section outlines the various options that can be set for the unit system used in a project. The project unit system may be accessed in the main simulator window by choosing **Project|Current Project Options** from the menu and selecting the **Unit system** tab, shown in the picture below.



Project unit system options

In the **Flow units** radio button group, you may choose from the following:

- cubic meters per day **(m^3/d)**
- litres per day (**L/d**)
- megalitres per day (**ML/d**)
- megagallons per day (**mgd**)

It should be noted that the same basis is used for flow and volume. For example, if you choose **L/d** as your flow unit, then volumes (e.g. for bioreactors, settling tanks) will be in litres also. Note that air flowrates are in units of m^3/hr.

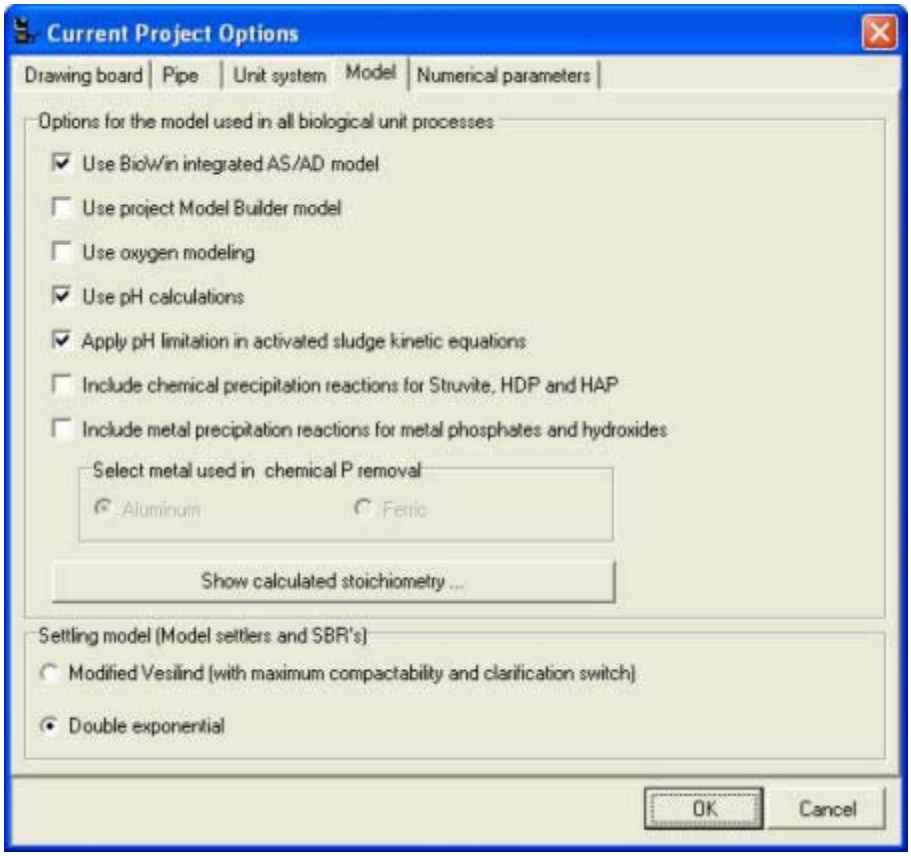
If you choose **mgd** as your flow basis, then the following imperial units will be used for other calculations in BioWin:

Measure	Imperial Unit
Length	feet (ft)
Area	square feet (ft^2)
Mass	pounds (lbs)
Pressure	pounds per square inch (psi)
Air Flow	Standard Cubic Feet per Minute
Specific Velocity	gal/ft^2/d
Mass Loading Rate	lbs/ft^2/d
Concentration	milligrams per litre (mg/L)

In the **BOD basis** radio button group, you may choose the length of time that BioWin uses to calculate BOD values. You may choose between **5**, **7**, and **20** day BOD.

Model Options

This section outlines the various project model options that can be modified in order to remedy any solution problems and improve simulation speed. These parameters may be accessed in the main simulator window by choosing **Project|Current Project Options** from the menu and selecting the **Model** tab, shown in the picture below.



Current project model options

There are a number of options for the model used in biological processes:

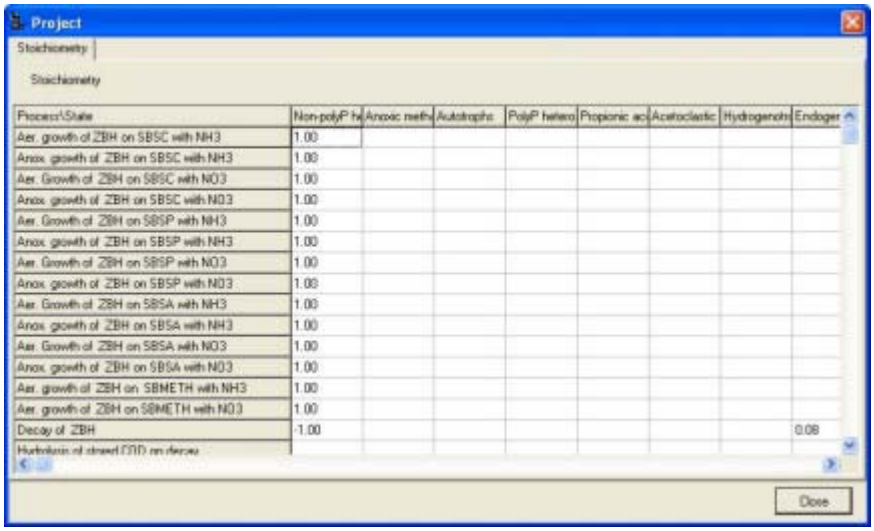
- You can choose whether or not you want to use the BioWin activated sludge process / anaerobic digestion model. Likely the only time that you would un-check this box is if you want to run another model in the biological processes (e.g. one of the IWA models). For details on the BioWin integrated ASP/AD model, please refer to the **Activated Sludge Model** and **Anaerobic Digestion Model**topics in the *Process Model Formulation* chapter.
- You can choose whether or not you use the current project's Model Builder model. If the BioWin model is unselected and the Model Builder model is selected, then only processes specified in the Model Builder model will be used. If both the BioWin and Model Builder

models are selected, then the Model Builder model will act as an "overlay" to the BioWin model. For information on using the Model Builder, please see the **Model Builder** topic in the *Element Types* chapter.

- Using the provided check box, users can select whether or not to use oxygen modeling. When this option is selected BioWin will model the impact of DO transfer in streams (from unit to unit and in the recycles) and the DO concentration in bioreactors when aeration is specified in terms of an airflow rate rather than a setpoint DO concentration. This selection impacts execution times, particularly for dynamic simulations. The following points should be noted regarding this option:

    1. If DO modeling is *not* switched on, and DO is specified by setpoint concentration in bioreactors, BioWin still computes oxygen uptake rates and other aeration features such as SOTR, OTR, SOTE, OTE, and air flow rate.
    2. If DO modeling is *not* switched on, and an air flow rate is specified for a bioreactor, the user is warned to switch on the DO modeling option so that the DO concentration in the bioreactor is calculated.
    3. If the user specifies a DO concentration for an input element and DO modeling is switched *off*, BioWin will ignore the specified DO concentration (i.e. the DO value will be taken as zero).
    4. If a DO value is entered for an input element and DO modeling is switched *off*, the DO value will not be changed in the file (even if the file is saved while DO modeling is switched off).

- Many of the process rates in the BioWin ASP/AD model are impacted by pH. If you would like to model pH inhibition in the model, you can do so by selecting the option in this dialog box. If you do not want to account for pH impacts on model rates, un-check this option.
- BioWin has the ability to model precipitation of Struvite, HDP, and HAP. If you would like to model these reactions, this box should be checked. For details on the modeling of these reactions, please see the **Spontaneous Chemical Precipitation** section of the *Process Model Formulation* chapter
- You can select whether or not to model chemical phosphorus precipitation reactions. If you select to model chemical phosphorus precipitation reactions, be sure to select the metal (i.e. iron or aluminum) to be used in the precipitation reactions. Note that the metal selected here will impact the interface shown in the **Metal Addition Influent**.
- You can view the stoichiometry of the current model by clicking the **Show calculated stoichiometry…** button. This button opens a window that shows the stoichiometry of the BioWin biological model that will be used at your current parameter values:



Stoichiometry of BioWin biological model at current parameter values

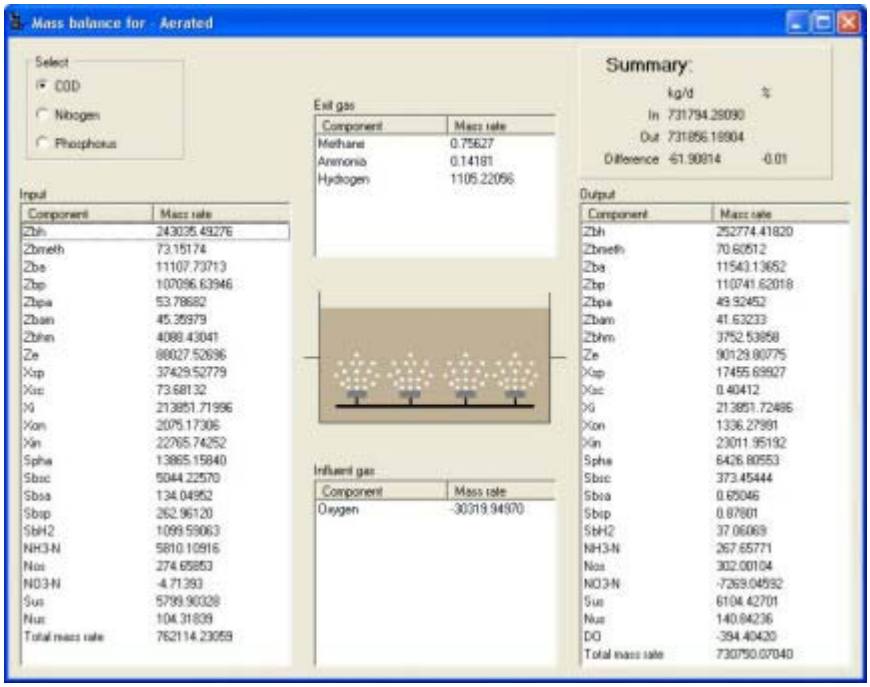
- Finally, this dialog box allows you to choose between BioWin's two available flux theory based settling models – a Modified Vesilind model or the well-known Double Exponential model. Note that the model selected here only is active in model secondary settlers and SBR units (i.e. this choice does not impact the operation of point and ideal secondary settlers). For more information on the Modified Vesilind and Double Exponential secondary settler models, please see the **Settling Models** section of the *Process Model Formulation* chapter.

You can also get model-related information for any particular element in a configuration. Right-clicking on an element and choosing either **Mass balance…** or **Rates…** displays the following model information.

Mass Balance Window

BioWin allows the user to see the mass balance of chemical oxygen demand (COD), nitrogen (N) and phosphorus (P) over any particular element. This window allows the user to select the material of interest (COD, N or P) and view the mass flow rates in and out of the vessel. At steady state conditions the mass balance should be zero, for each of these items, that is the amount of COD, N or P flowing should be the same as the amount flowing out of the vessel. The figure below shows an example of a mass balance window for a bioreactor at steady state conditions.

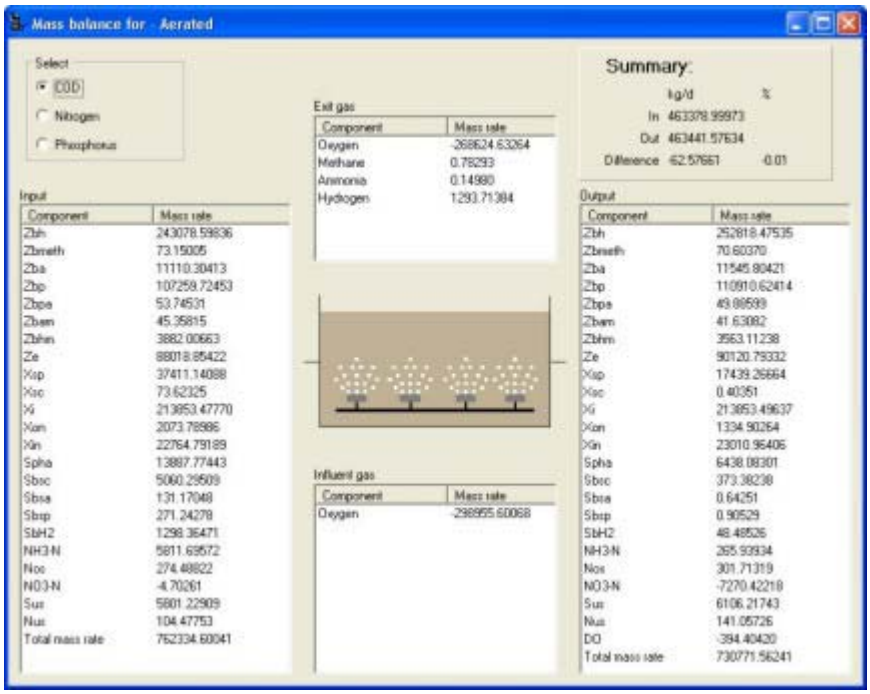
Note: Different elements may have slightly differing layouts for the mass balance window. For example a clarifier element (which has no gas input uses the list box below the graphic to represent the underflow.



Mass balance window for a bioreactor at steady state conditions

The list box on the left represents the liquid input to vessel and the list box on the right the output from the vessel. The gas phase is represented by the list boxes above and below the graphic.

Note: If the gas phase is not modeled then BioWin reports the amount transferred to or from the liquid phase rather than the actual gases stripped from the liquid phase (in COD units). In the figure below the gas phase was modeled and consequently the oxygen reported in the lower list box represent the entire oxygen content of the gas, and the majority of the oxygen is also reported in the exit gas (top list box).



Mass balance window for a bioreactor at steady state conditions with gas phase modeled

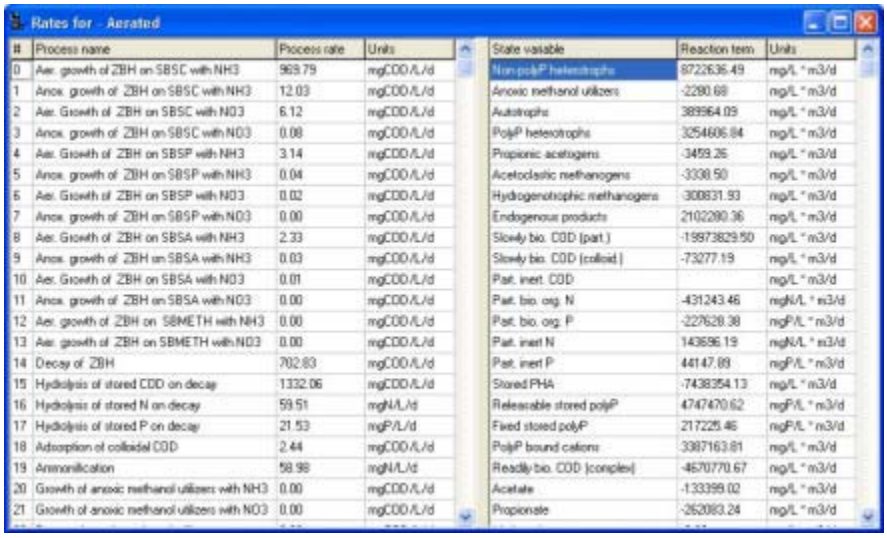
In each of the list boxes, any state variables that contribute to the material selected (COD, N, or P) are listed and the mass flow rate for each given. You will notice that although the figure above shows the mass balance for COD the nitrate (NO3-N) state variable is listed in both the input and output list boxes and that it is negative. This is because nitrate can be used as an electron acceptor. The mass balance window takes this into account and therefore reports NO3-N as a negative number. The mass balance window also accounts for the oxygen demand associated with ammonia so it too is reported in the list box.

Note: If the COD mass balance is selected then all items in the list box (including nitrogen components) are reported on COD units.

Note: If nitrogen precipitates are present they may also be listed in the mass balance window for COD and N.

Rates Window

BioWin allows the user to view the rates of reactions in bioreactor vessels. The rates window will normally be used only as a diagnostic tool for simulation difficulties. The rates window shows the rate (mg L^{-1} d^{-1}) for each of the processes in the BioWin biological model and the reaction term (g d^{-1}) for each of the state variables. An example of the rates window is shown below.

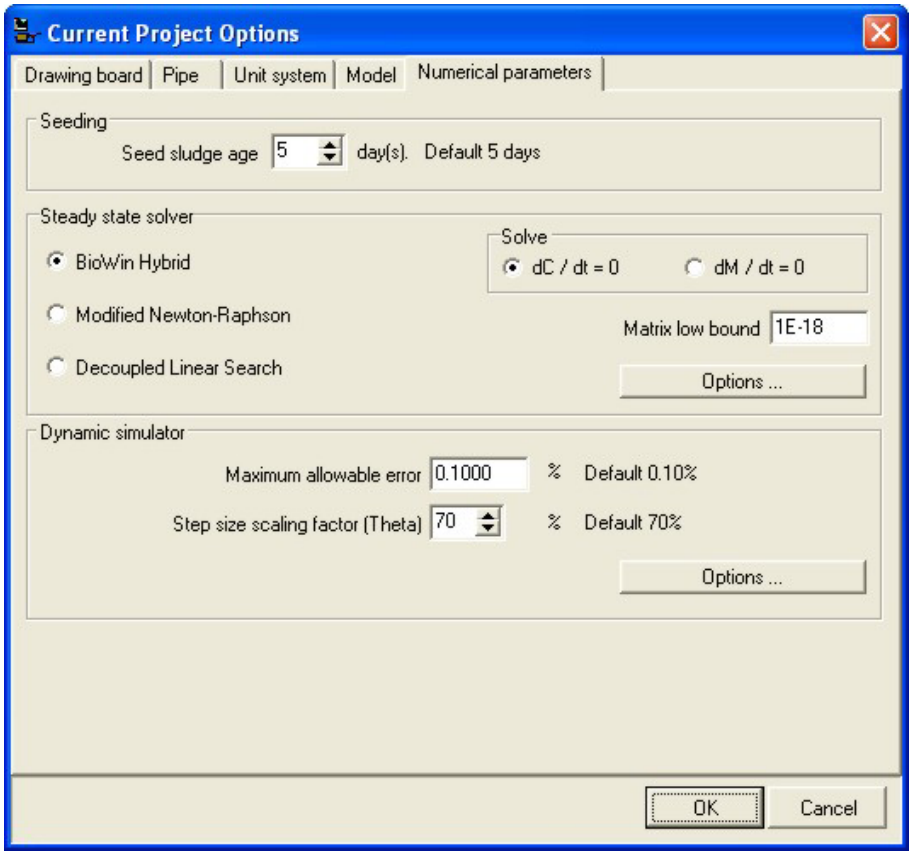


Rates window

Note: The units of the reaction term are always in g d^{-1} regardless of your current project unit selection.

Numerical Parameters

This section outlines the various project model options that can be modified in order to remedy any solution problems and improve simulation speed. These parameters may be accessed in the main simulator window by choosing **Project|Current Project Options** from the menu and selecting the **Numerical Parameters** tab, shown in the picture below.



Current project numerical parameters

The **Seed sludge age** parameter is used to calculate seed values for the biological system. The seed values help the BioWin solver engine to determine an appropriate solution for the system, or are used at the beginning of dynamic simulations (if you select to start a dynamic simulation from seed values).

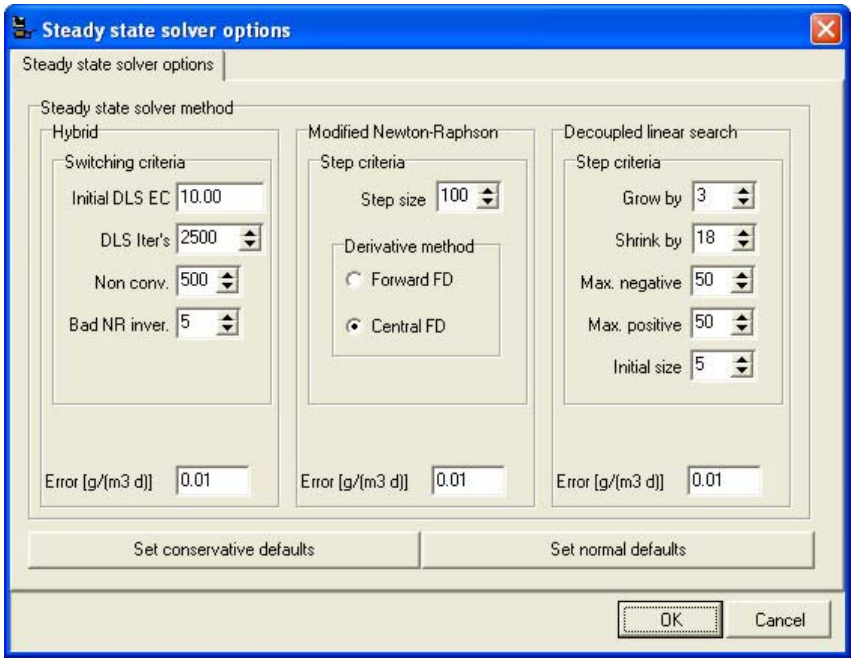
BioWin uses the seed sludge age that you specify, along with the strength of the influent wastewater and reactor volumes, to perform an approximate steady state calculation in order to guess at what the concentrations will be in elements. The analysis is based on simple continuous flow steady state equations, so it only is approximate. Note that the seed sludge age in no way affects the actual SRT of a plant configuration - this is set strictly by the mass of solids that are removed from the system each day (i.e. via wastage and decant).

If you suspect an inappropriate solution, or if BioWin has trouble finding a steady state solution, the seed sludge age may be changed in order to provide different seed values. These different seed values may improve the solution behavior.

The **Steady state solver** is a powerful search algorithm provided in BioWin to solve the model under steady-state (constant flow, concentration and operating) conditions. The **BioWin Hybrid** method utilizes a combination of the **Newton-Raphson (NR)** second order search, and a **Decoupled Linear Search (DLS)**. Both algorithms have advantages in certain situations, and BioWin will select the best method and switch between them if necessary. **Newton-Raphson** prefers smooth, continuous functions, and can reduce the error by two orders of magnitude in an optimal case. The **DLS** method uses much smaller steps and is not sensitive to sharp discontinuities and non-linearities.

Steady State Solver Options

The user can select from the three available methods for steady state convergence. There also are a number of options (available by pressing the **Options** button) for the each of the different methods. Key options are explained here.



Steady state solver options

Hybrid:

Initial DLS EC (Decoupled Linear Search Error Criteria) (10) - BioWin will use the **DLS** method until it achieves an error of less than the initial **DLS** error criteria **or** it performs more than the initial **DLS** iterations [**DLS Iter's" (2500)**] at which time it will switch to the **NR** method.

Non conv. (500) - This value is used to determine if the **NR** solver is stuck. Increasing it will make the solver consider itself stuck over a wider range of values.

Bad NR inver. (5) - If the **NR** solver takes this many steps in the a direction which results in an increase in the error then **DLS** will be activated

Modified Newton-Raphson:

Step size (100%) - Use the original **NR** estimated steps for each state variable. Can be reduced to make the method more cautious.

Derivative Method - This setting can be used to choose between central finite difference and forward finite difference numerical derivative approximations.

Decoupled Linear Search:

Grow by (3) - Step size increasing by 0.03% for successful iterations.

Shrink by (18) - Step size decreasing by 1.8% for failed iterations.

Max negative (50%) - Maximum allowed negative step.

Max positive (50%) - Maximum allowed positive step.

Initial size (5%) - Size of initial perturbation.

Error [0.01 (units of mg/L/d {g/m^3/d} if using the default dC/dt and g/d if using dM/dt)] - Error criteria to stop iterations if mass balance is closer than this to the solution.

Note: You can set the steady state solver to use conservative parameter values if you have trouble finding a steady state solution for a configuration (see **Tips for Systems that are Difficult to Solve** below).

Dynamic Simulator Options

There are also options for the **Dynamic Simulator.** Increasing the **Maximum allowable error (0.1%)** may improve dynamic simulation speed, but the reduced error criterion may also result in poorer mass balances. The **Step-size scaling factor (70%)** determines how the dynamic solver adjusts the step size.

There also are a number of options (available by pressing the **Options** button) for fine tuning the simulator performance. You can also specify the **Minimum size for relative error calculation (0.0001)** which is the value below which error calculation will switch to absolute error.

The **Excessive negative rate damping** section contains the setting to control the PowerMonod. Traditionally Monod switching functions are used in activated sludge models to turn rates on and off depending on concentration of oxygen, substrates, nutrients, etc. In chemical systems however, the rates and functional concentration changes cover many orders of magnitudes. A standard Monod switching function provides about one order of magnitude reduction in the rate if the concentration decreases one order of magnitude. In Chemical systems this is frequently insufficient to effectively shut down a chemical rate. This may result in an excessively slow dynamic simulation when a high negative rate is affecting a small concentration as the solver has to dramatically reduce the integration step size to prevent negative concentration values. In BioWin, under special circumstances, a more powerful switching function, (PowerMonod) is used to turn off precipitation rates based on the available soluble metal concentration. The PowerMonod expression is only applied during dynamic simulations.

Concentration limit (default = 0.001 mg/L) - The PowerMonod function is only applied below this concentration limit. This approach is in contrast to the traditional Monod switching functions that are applied at all concentrations. Reducing this concentration limit allows increased accuracy, but the simulation speed is impacted. The concentration limit should never be increased beyond its default value (0.001 mg/L).

Damping power (default value = 3) - This is an exponent applied to the Monod saturation function. The traditional Monod expression has a damping power of 1. The amount of damping increases as the Damping power is increased. In typical precipitation runs the default should provide satisfactory results.

Under extreme pH values it may be necessary to increase the Damping power (to 5 or 10) to improve simulation speed.

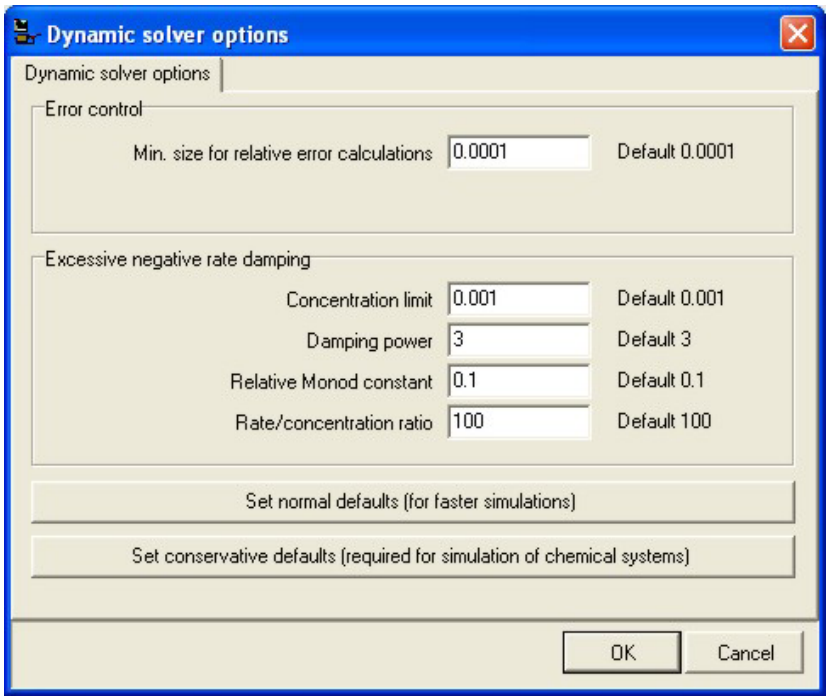
Relative Monod constant (default value = 0.1) - The PowerMonod expression is applied in the range from 0 to **Concentration limit**. This parameter behaves like a half saturation value within this range. The default value of 0.1 is equivalent to 10% of the **Concentration limit** (0.1 * 0.001 = 0.0001 mg/L at the default values). Typically this parameter does not require adjustment.

Rate/concentration ratio - The PowerMonod function will only be applied if:

- the calculated rate is negative, and

- the absolute ratio of the calculated rate to the concentration is larger than this parameter.

Typically this ratio normally does not need to be adjusted.



Dynamic simulation options

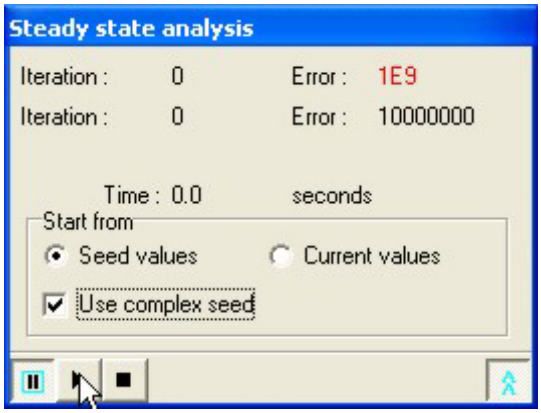
Tips for Systems that are Difficult to Solve

By default BioWin uses an adaptive hybrid solver to search for a steady state solution. This solver uses a combination of a modified Newton-Raphson method and an uncoupled search algorithm. Typically the solver will find a solution in less than 15 iterations, although for complex systems this might take longer.

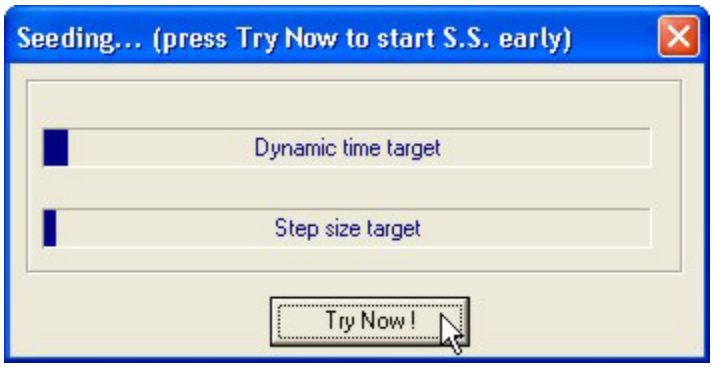
Note: After a system has been solved to a steady state solution it is almost always easier for the solver to converge on the solution from current values rather than from seed. If a particular organism type is "washed-out" in a steady state solution then starting from current values may bias the solver against them and it may be better to start from seed..

If the solver is experiencing difficulties in finding a solution the parameters uses in the solver are adapted automatically, nevertheless there may be situations where either the solver is still unable to determine the solution, or the user wishes to changes the solver parameters to improve performance. [The solver does not improve the error value for more than 20-30 iterations, or becomes very slow.] If you are experiencing difficulties in determining the steady state solution for a particular configuration you have a number of options.

1. Check that volumes, flows, DO settings, influent concentrations are all valid. Try matching the seed sludge age of the system. If you are using SRT control, try fixed wastage flow first.
2. Try using the complex seeding method (non SRT control systems).



Start your simulation. [Press **F6** or select **Simulate | Steady State**] This will open the **Steady State solver** dialog box, check the **Use Complex seed** checkbox before pressing the **Run** button (as shown below).



Press the **play** button to begin the simulation. BioWin opens a dialog which shows the progress of the complex seeding. This dialog is shown below.

The complex seeding essentially performs a dynamic simulation at the steady state conditions. The dialog box will close automatically if either the **Dynamic time target** or the **Step size target** is achieved. Generally it is not necessary to wait for the dialog box to close automatically as a short period of dynamic seeding will usually be sufficient to improve the performance and the user can press the **Try Now!** button at any time to start the main solver.

3. A third approach is to try using more conservative settings for the solver. BioWin provides a set of **Conservative** parameter defaults (see **Steady State Solver Options**above) or you can enter your own selection of parameter values.

Note: Using the conservative values forces the solver to make smaller adjustments which usually increases the number of iterations but also stops very bad adjustments. These values are stored with the project file, and do not affect other simulations.

Note: The BioWin default **conservative** values are particularly applicable to systems in which spontaneous chemical precipitation (HDP, Struvite) is probable as these reactions have extremely rapid reaction rates.

4. For certain systems it may be better to use only one of the solution techniques. That is rather than using the hybrid method, use either the modified Newton-Raphson method or the search algorithm. The **Numerical Parameters** tab allows you to select any one of these methods.

Note: The double exponential settler model is generally not well behaved for the Newton-Raphson method.

5. Simplify the model.

   Simplify the model by turning off one (or more) of the options on the **Model Options** tab; namely

   - pH inhibition for activated sludge kinetics.
   - pH calculations.
   - Oxygen modeling.
   - Chemical precipitations.
   - Metal precipitation chemistry.

   Now try to solve the system, if a solution is achieved then you may start turning the options back on and resolving starting from the **Current values**.

6. For very complex configurations with difficulties finding a solution a general step-wise method may be useful:

   - Simulate with pH modeling, limitation, etc turned off.
   - Enable pH modeling, simulate from current values.
   - Enable pH limitation, simulate from current values.
   - Enable Struvite, Ca-phosphates modeling (if required), simulate from current values.
   - Enable metal precipitation modeling(if required), simulate from current values.

   Repeat this step-wise method with conservative settings solver settings, or using only one of the solver method.

7. If a configuration does not converge in spite of trying the methods described above:

   - Consider simplifying the configuration if at all possible.
   - Send the configuration (*.bwc file) with the description of the problem (exact steps) for support to **info@envirosim.com**)

Note: Under some circumstances merely stopping and then restarting the solver from **Current values** will assist the solver. This essentially resets the solver parameters, but provides different seed values.

Note : The **Solution not found** message is displayed when the solver has :

- Identified a singular matrix – in which case you may be able to solve using the selecting the Decoupled Linear search option on the **Numerical Parameters** tab.
- Exceeded the maximum number of iterations allowed – you should probably try one of the methods described above.
- Experienced numerical instability or range problems – you should probably try one of the methods described above.

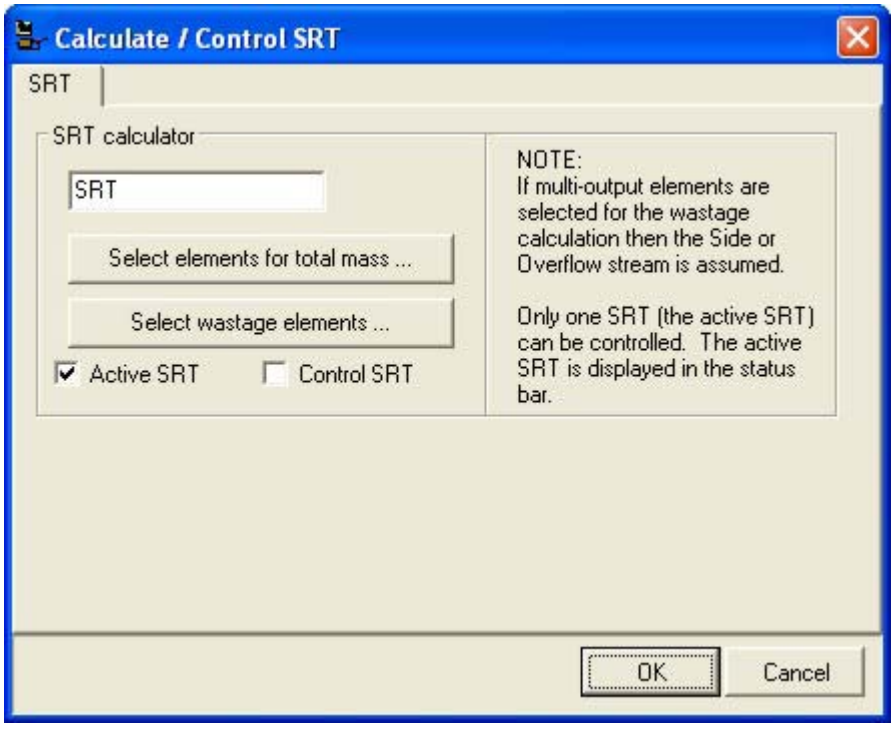
Solids Retention Time Calculation

SRT tool

BioWin offers the functionality of Solids Retention Time (SRT) calculation for a project process configuration. The dialog box shown below, accessed via the **Project|Active SRT** menu command or the calculate/control SRT tool, is used to supply BioWin with the information required to perform this calculation.

An enhancement that has been made regarding SRT calculation is that BioWin now has the ability to store multiple SRT calculation scenarios. For example, suppose that you want to look at the impact of effluent suspended solids on the SRT of a system. To do this you could set up one SRT calculator that includes the effluent as a wastage element, and one SRT calculator that does not. Another possible scenario would be if calculating the SRT for each sludge in a two sludge system.

The first step in setting up an SRT calculator is to select the **Project|Active SRT** menu command or to click the **New…** button on the calculators toolbar. When you do this, you will be presented with the following dialog box.

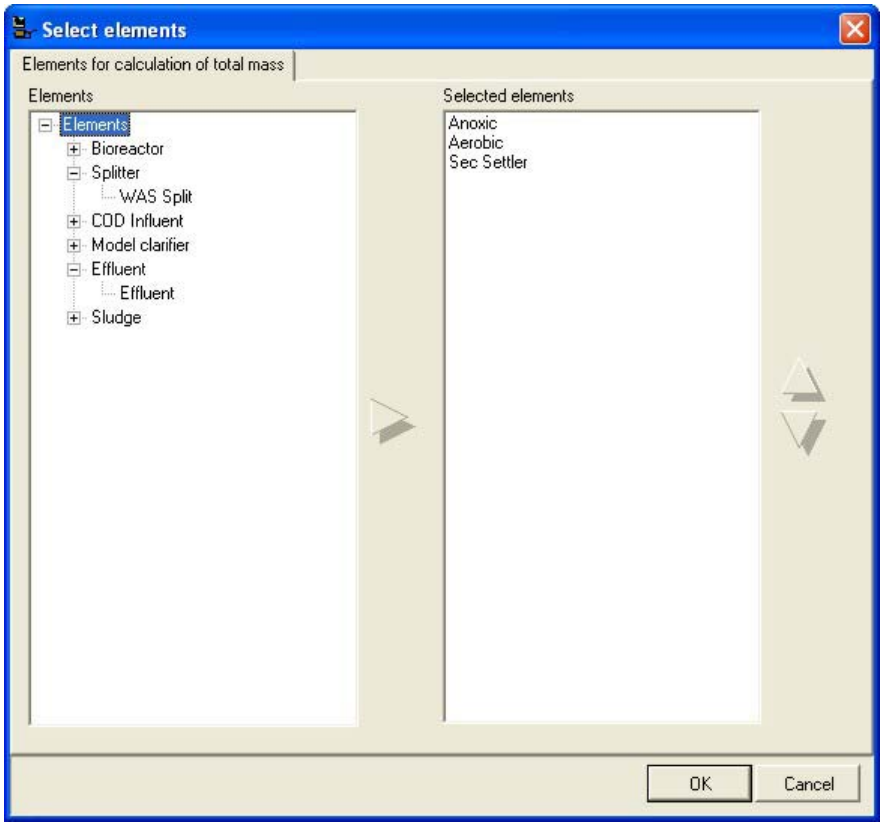


Project SRT calculation dialog box

Enter a name for this SRT calculation scenario in the text box provided. You can make this SRT the **Active SRT** by clicking the check box with this label. Next, you

must specify the elements in the configuration that will be used to calculate the total mass in the system. To do this, click the **Select elements for total mass…** button. When you do this, you will be presented with a two-paned dialog box, shown below.

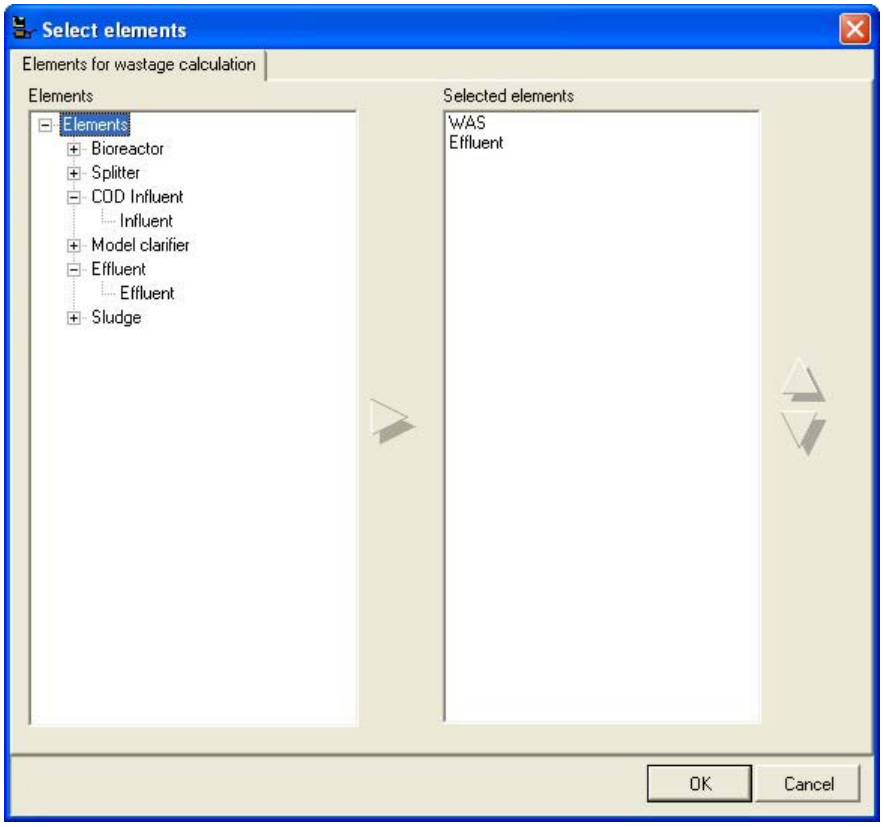
Note: If you have set up more than one SRT calculator, the active SRT will be displayed in the main simulator window status bar. To change the SRT calculator that currently is active, go to the calculators toolbar, select the SRT calculator you want to make active from the drop list, and click the **Configure…** button on the calculators toolbar.



Dialog box used to select elements for total mass in SRT calculation

In the left-hand **Elements** pane, there is an expandable/collapsible tree view of the various elements in the configuration. You can expand/collapse the branches of this tree by clicking on the +/- signs next to the branches, or by double-clicking on the branch labels. Using this navigation technique, locate the elements that you wish to include in the total mass calculation. When you find an element that you wish to include, move it to the right-hand **Selected elements** pane by clicking on the element name and then clicking the right-pointing arrow located between the two panes. When you have added all the elements that you wish to include in the total mass calculation, click the **OK** button.

The next step in SRT calculation is to select the elements in the configuration from which wasting of solids is taking place. To do this, click the **Select wastage elements…** button. When you do this, you will be presented with another two-paned dialog box, shown below.



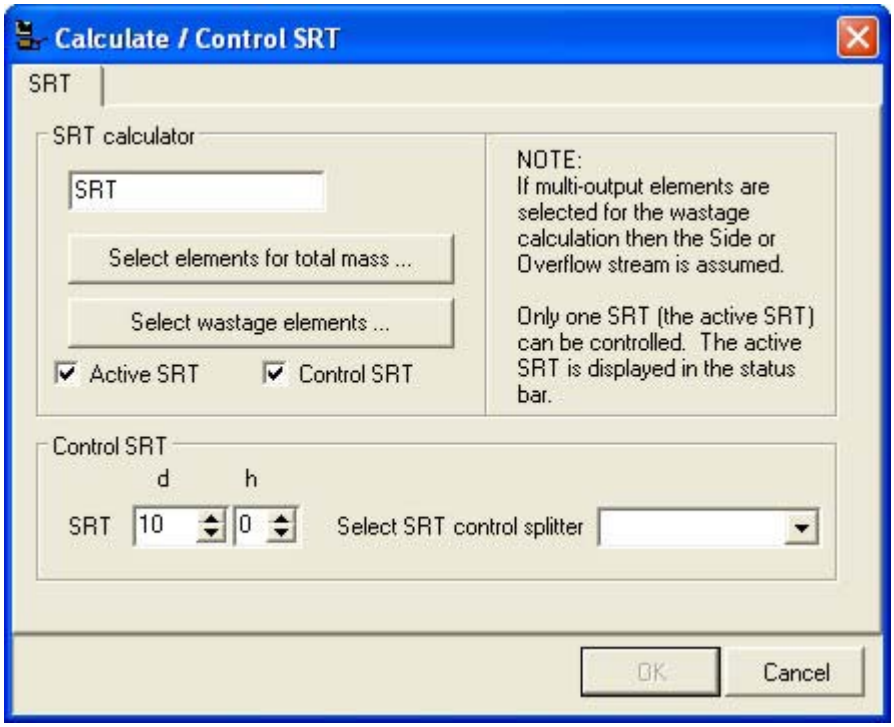
Dialog box used to select wastage elements for SRT calculation

In the left-hand **Elements** pane, there is an expandable/collapsible tree view of the various elements in the configuration. You can expand/collapse the branches of this tree by clicking on the +/- signs next to the branches, or by double-clicking on the branch labels. Using this navigation technique, locate the elements that solids are being wasted from. When you find an element that you wish to include, you move it to the right-hand **Selected elements** pane by clicking on the element name and then clicking the right-pointing arrow located between the two panes. When you have added all the elements that you wish to include in the total mass calculation, click the **OK** button. Note that if any multi-output elements (e.g. clarifiers which have underflow and overflow outputs) are specified as wastage elements, BioWin assumes that solids wasting is occurring in their Side or Overflow streams.

Now that you have completed the two required steps for SRT calculation, you may close this dialog by clicking **OK**. The next time that you perform a steady state or dynamic simulation, the main window status bar will be updated and the SRT in time units of days will be displayed. Note that the frequency of this update is governed by the summary pane refresh frequency.

Note: To remove an element (or elements) from the **Selected elements** column of either of the dialog boxes used to select elements for SRT calculation, click once on the element name so that it is highlighted blue, and press the **Delete** key on your keyboard.

Once you have given BioWin the necessary information it needs to calculate the SRT for a configuration, you can extend BioWin's SRT calculation functionality by telling BioWin to control the wastage of solids from a particular splitter element in order to achieve a desired SRT. To do this, check the box labeled **Control SRT**. The dialog box will now change in appearance to the one shown below.



Project SRT calculation and control dialog box

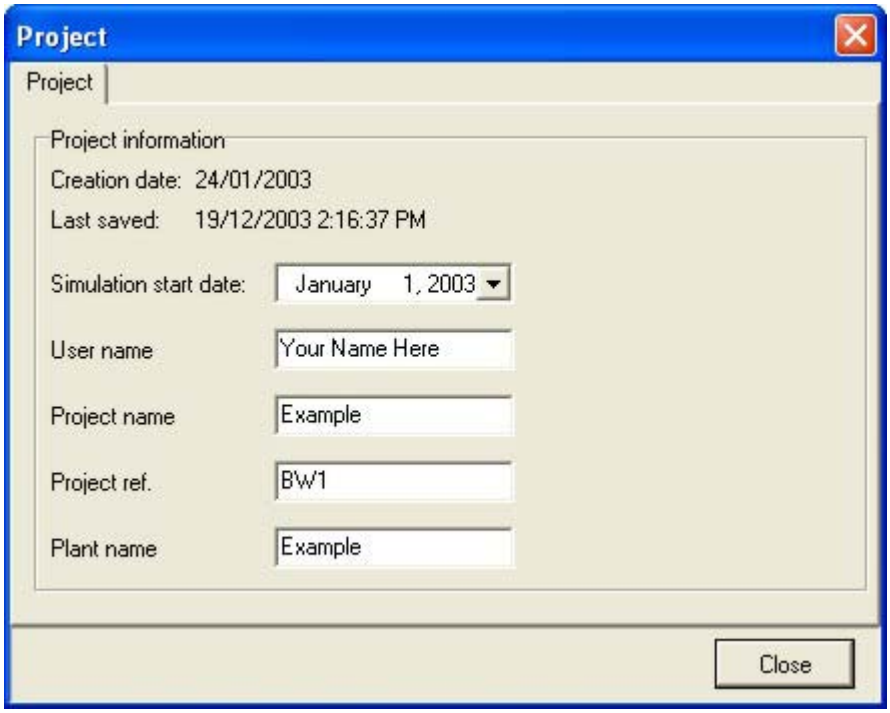
A **Control SRT** group now occupies the lower portion of the dialog box. Two spin edit boxes (one marked **d** for days and one marked **h** for hours) are used for setting the desired SRT. Using the **Select SRT control splitter** drop list box, you then may select the splitter element in the configuration that BioWin will use to control the wastage rate in order to achieve the desired SRT. BioWin assumes that the side stream of the splitter will be the wastage stream, and it changes the setting in the splitter element to control the side stream flow. When you click **OK** to close this dialog box, you will notice that the splitter that you specified as the control will have a new drawing board icon that shows a valve on the wastage stream. Note that you will be unable to close this dialog box with the **OK** button if you have not specified all the required information.

If you now perform a steady state simulation, BioWin will adjust the wastage rate out of the control splitter so that the SRT of the configuration is that which you have specified. This SRT will be displayed in the main window status bar once the steady state solution has been found. It should be noted that the SRT controller only sets the wastage rate for steady state conditions. If you begin a dynamic simulation after you complete a steady state simulation, the wastage flow rate will remain constant at the value calculated. That is, there is no control of wastage rates during a dynamic simulation.

Note: It is now possible to plot SRT against time. If you have multiple SRT calculators set up, you can plot as many of them as you wish. For information on setting up an SRT plot, see the **Special Series (from Album)** section of the *"Creating Charts and Adding Series"* chapter.

Specifying Project Details

This section outlines the various details that can be specified for the project. The project details may be accessed in the main simulator window via the **Project|Info** menu command, which will present you with the dialog box shown below.



Dialog box used for entering project details and information

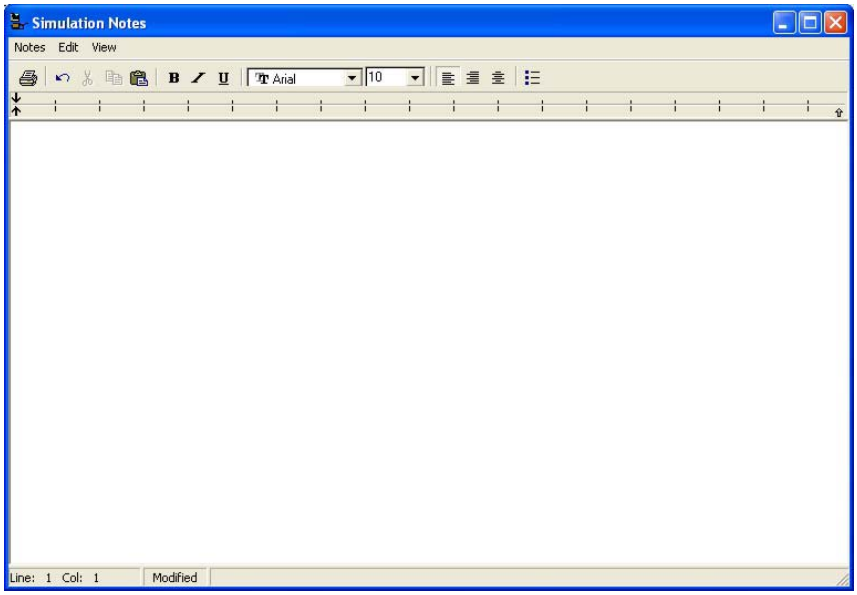
The **Project information** group contains details on the current project:

- **Creation date:** this field indicates the date and time that the current BioWin project was created.
- **Last saved:** this field indicates the date and time that the current BioWin project was saved.
- **Simulation start date:** use this field to set the date that will be used as the default starting date for dynamic simulations. Note that if you want to specify another date different from the one you specify here when doing a dynamic simulation, you may do so. When you click the down arrow you will be shown a calendar from which you can choose the date. The current day is circled in red, and the day that currently is selected is indicated by a solid blue oval. You may use the left and right arrow buttons at the top of the calendar to move to different months.
- **User name:** use this field to indicate the name of the person responsible for the project.
- **Project name:** use this field to indicate the name of the project.
- **Project reference:** use this field to indicate the project reference code.

- **Plant name:** use this field to indicate the name of the plant that the simulation project is for.

Recording Project Notes

The project note editor, shown in the picture below, is a tool that may be used to record information relevant to the current project. You can access the editor via the menu command **Project|Notes**. It is similar in operation to the WordPad™ utility that comes with Windows 95. The notes that you make can have some formatting applied to them since they are Rich Text Format files (*.RTF). When you save the BioWin project that the notes are associated with, the notes will be saved as a separate notes file (*.NTS) that can be accessed by other applications such as word processors since it is in standard rich text format. Note that the notes will always be available in BioWin when the project they are associated with is active.



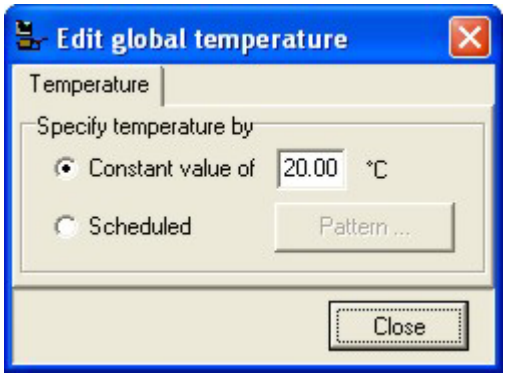
The project notes editor

Specifying Project Model Parameter Values

BioWin allows you to override the default model parameter values for a project via the **Project|Model Parameters** command. This command invokes the **Model Parameters** editor. For information on this dialog box, see the **Model Parameter Editor** section. The changes made to project model parameters using this tool will affect all elements in the project configuration except those that have local parameters specified.

Specifying Project Temperature

BioWin allows you to specify a global temperature for a project via the **Project|Temperature** command. Invoking this command presents you with the **Edit global temperature** dialog box, shown below.



Dialog used to set the global project temperature

In the **Specify temperature by** radio button group, you may choose from two options for global temperature. If you want a constant temperature in your project elements, then enter the desired temperature value in the **Constant value** of text edit area. Note that if you wish, you can override this global temperature in individual elements.

If you want a time-varying temperature pattern, then select the **Scheduled** radio button. This will activate the **Pattern…** button. Clicking this button presents you with the **Edit temperature itinerary** dialog box.

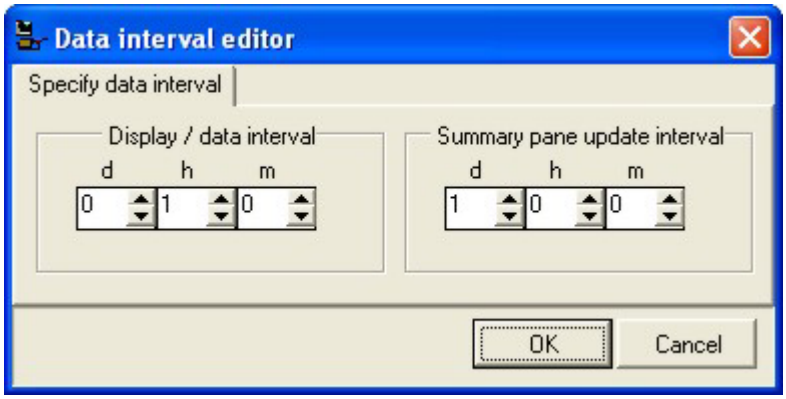
Note: Use care when switching back and forth between constant and scheduled temperature values. BioWin assigns the constant value to the first value of your schedule, and vice versa. So in the case where you are switching between a constant value that is *different* from the first value of your scheduled pattern, ensure that these values are what you want them to be before running simulations.

Managing Data

Data management is an important component of BioWin projects. This section covers topics related to the database BioWin uses to store simulation-generated and imported data for a project. Data options and manipulation are accessed via the **Project|Database** menu in the main simulator window, and the **Database** menu in the album.

Data Interval

BioWin allows you to set data monitoring intervals on a project-to-project basis, via the **Project|Data interval** command. This will present you with the **Data interval editor**, shown below.



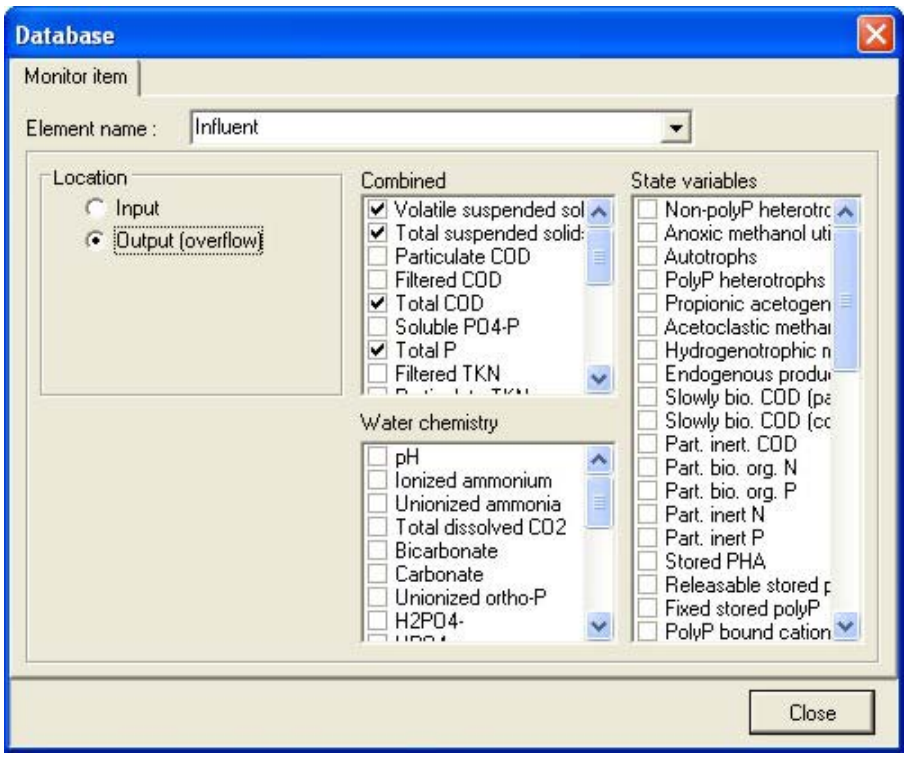
Project data interval settings

There are two data intervals that you may set if you wish, using the appropriate spin edit boxes. The first is the **Display / data interval**. This is the data interval that will be used by BioWin to log data into the database, and this value also will be used as the point drawing interval on charts and the data refresh interval for tables and element information displays.

The second is the **Summary pane update interval**. This is the time interval that BioWin will use to refresh the main window summary panes and status bar. It should be noted that if this value is set to a very low number (i.e. high refresh frequency), it might result in decreased simulator speed because of the necessity to redraw the contents of the main window summary pane. If you don't wish the summary panes to have any automatic refresh, enter a value of zero in each spin edit box. One final point should also be mentioned. Regardless of the value that you enter, the contents of the main window summary pane may be refreshed by holding the mouse cursor over an element, as long as the summary panes are in **Fly by** mode.

Monitoring Data

You have complete control over the project simulation data that are logged to the database. This allows you to optimize the size of the project database so that you only record what you are interested in. Selecting **Project|Database|Monitor item…** will present you with the dialog box shown below used for specifying the variables/parameters you want logged to the database.



Dialog used to set up monitoring for elements in a project configuration

Select the element you wish to monitor an item for from the **Element name** drop list box. The **Location** radio button group is used to choose the location in the element (i.e. Input, Output {overflow}, Underflow) where you wish to obtain the data to be logged. Note that this group changes depending on the type of element you have selected. For example, the underflow location only is shown for elements that have an underflow such as settlers. For a selected element, you may choose the parameters/variables you wish to monitor using the **Element specific**, **State variables**, or **Combined** check list boxes. Like the **Location** group, the **Element specific** list is dependant on the type of element selected.

Another method to set up monitoring for an element is to use the **Monitor items** tab. You can access this tab by right-clicking on the element in the drawing board and selecting **Properties…** from the resulting popup menu, or by double-clicking on the element. Use of this tab is identical to that of the **Project|Database|Monitor item…** dialog, except for the fact that you don't select an element (you've done this by clicking on the element in the drawing board).

If you find that you are always monitoring the same items in the same types of elements from project to project, you may want to use BioWin's **Auto-logging** feature. You can use this feature to tell BioWin to always monitor certain parameters/variables on an element-type basis. For more information, please see **Automatic Logging Options**.

Un-monitoring Data

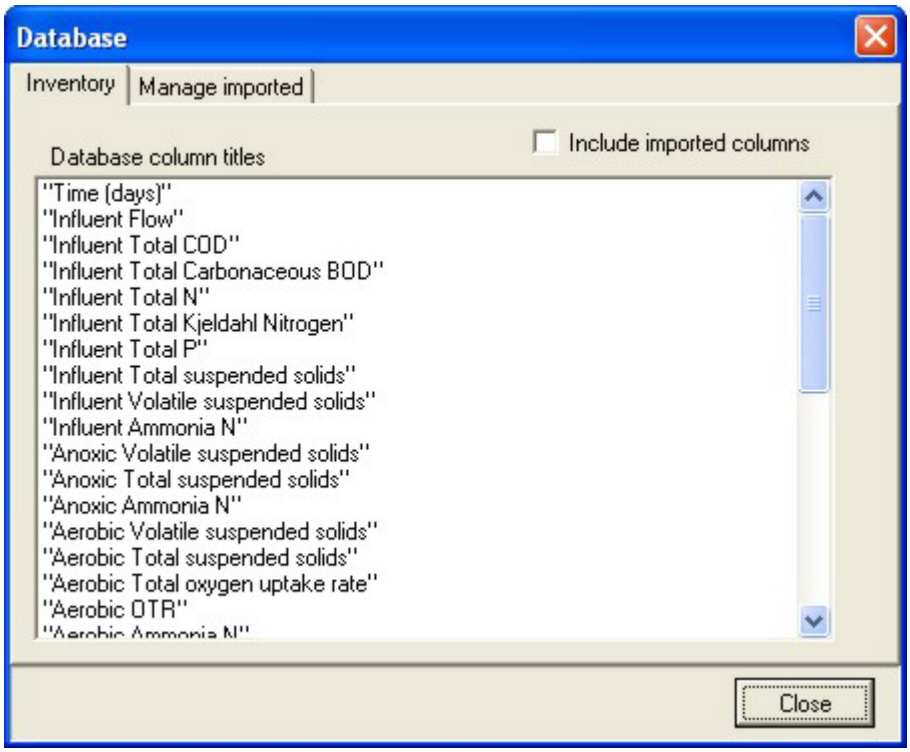
BioWin now offers the functionality of un-monitoring state variables and parameters. You can un-monitor an item by following the procedure outlined for monitoring items (except that you un-check boxes for parameters that had been previously monitored). This gives users increased control over what is logged to the BioWin database. The following points regarding un-monitoring should be noted:

- When you plot a time series chart, BioWin automatically monitors the variables / parameters that you plot. If the chart is deleted later, the items will be un-monitored.
- If you set up monitoring for items following the procedure outlined above, create a chart of these items, and then delete the chart, **the items will still be monitored**.
- If you set up monitoring for items following the procedure outlined above, create a chart of these items, and then attempt to un-monitor the items, BioWin will not allow the items to be un-monitored. That is, the items will still be monitored because this is required for the chart(s) using them.

Note : if you delete the chart, the items will be un-monitored, i.e. BioWin "remembers" your disallowed attempt at un-monitoring and carries out the request when the chart(s) using the items are no longer present.

Database Inventory

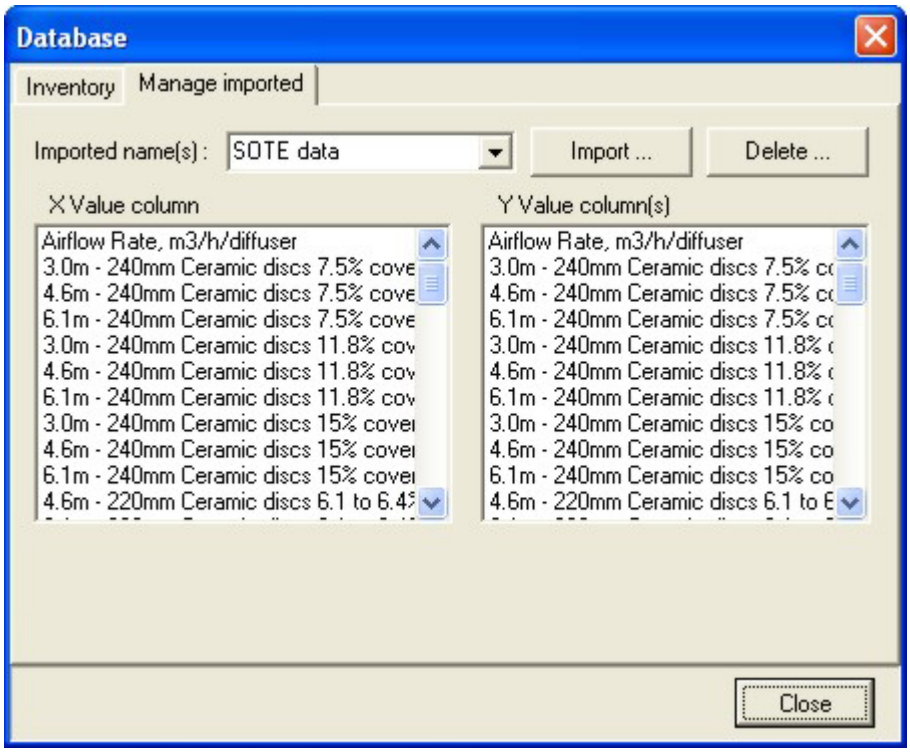
BioWin offers a means to display what is being monitored to the project database, as well as manage data that have been imported. Selecting **Project|Database|Inventory list…** will present you with the inventory dialog box, shown below.



The database inventory list of monitored items

This dialog lists the **Database column titles**. Since these are named with the convention "**lementName MonitoredParameter**, by looking at the list you can see what is being logged to the database. If you select the box labeled **Include imported columns**, the list also will show the column titles of any imported data that are contained in the database.

This dialog box also gives you the facility to manage data that you have imported into a BioWin project. Clicking the **Manage imported** tab reveals the following dialog box.



Dialog box used for managing imported data

You can use the **Imported name(s)** drop list box to view the contents of the various data files that have been imported to the BioWin project database. When you select a file, the column names are displayed in the **X Value column** and **Y Value column(s)** lists. This is useful to verify that the file you have imported contains the data you are interested in. If you decide that you no longer want the file to be included in the database, clicking the **Delete…** button will remove it.

If you wish to import a data file into the database, clicking the **Import…** button will launch the procedure for doing so. You will be presented with the **Open File** dialog box. Once you have selected the file you want to import, the **Import Wizard** will take you through the remaining steps.

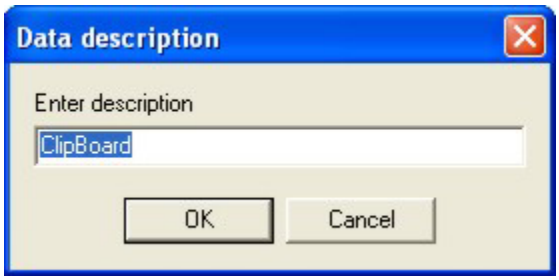
Importing Data

BioWin offers two methods for importing data to the project database. You may import data from:

1. A file, or;
2. The clipboard if you have copied a selection.

If you choose to import data from a file via the **Project|Database|Import file…** command, you will be presented with the **Open File** dialog box. Once you have selected the file you want to import, the **Import Wizard** will take you through the remaining steps.

If you choose to import data from the clipboard via the **Project|Database|Import clipboard…** command, you will be presented with a dialog box that allows you to enter a brief description of the data, shown below.



Dialog box for entering description of data imported from clipboard

This dialog gives you a chance to assign a descriptive name to the data that you are importing from the clipboard. This name will be associated with the block of data so that you recognize it when you attempt to use the imported data in other BioWin operations, such as plotting. This option is not required for file imports as the file name is used to identify the data. Once you have provided the clipboard data block with a name (or accepted the default name – do this with caution as you may end up with multiple identically-named imported data), the **Import Wizard** will take you through the remaining steps.

Exporting Data

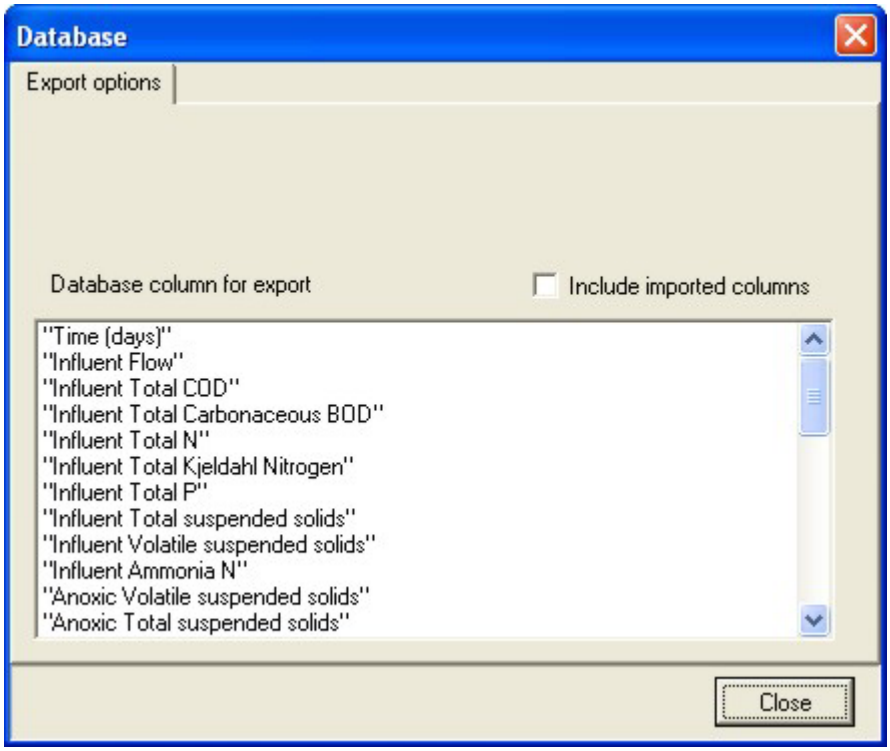
BioWin offers two methods for exporting data from the project database. You may export data to:

1. A file, or;
2. The clipboard.

If you choose to export data to a file via the **Project|Database|Export file…** command, you will be presented with the **Save File** dialog box. You may choose to export to one of the following file formats:

1. Text file (*.txt) – in this format, columns of data will be delimited by tabs, which is a suitable format for import to a spreadsheet application.
2. Comma-separated values (*.csv) – in this format, columns of data will be delimited by commas. This file format also may be imported to a spreadsheet application.
3. All file (*.*) – in this format, columns of data will be separated with a delimiter of your choice.

Once you have used the **Save File** dialog box to specify a name, file type, and location for you exported data file you will be presented with the following dialog box:



Dialog box showing data to be exported to file

A list shows the database column titles that will be exported. By checking the box labeled **Include imported columns**, you can also export to this file any data that you have previously imported to the database. Note that the **Specify delimiter for export** radio button group only will be available if you have selected the third file format from the above list.

If you choose to export data to the clipboard via the **Project|Database|Export clipboard…** command, you will be presented with a dialog box identical to the one shown above with the exception that no choice of delimiters is offered. Tabs will separate the data columns you paste to the clipboard since this is the best format for copying the data into a spreadsheet application.

Custom Export Utility

BioWin offers you an increased level of control over what is exported from the database, via the **Database|Custom export file…** and **Database|Custom export clipboard…** commands.

These commands are different from the "non-custom" export database commands in that they allow you to select which elements you want to export data for, the parameters that you want to export, and the order of the parameters.

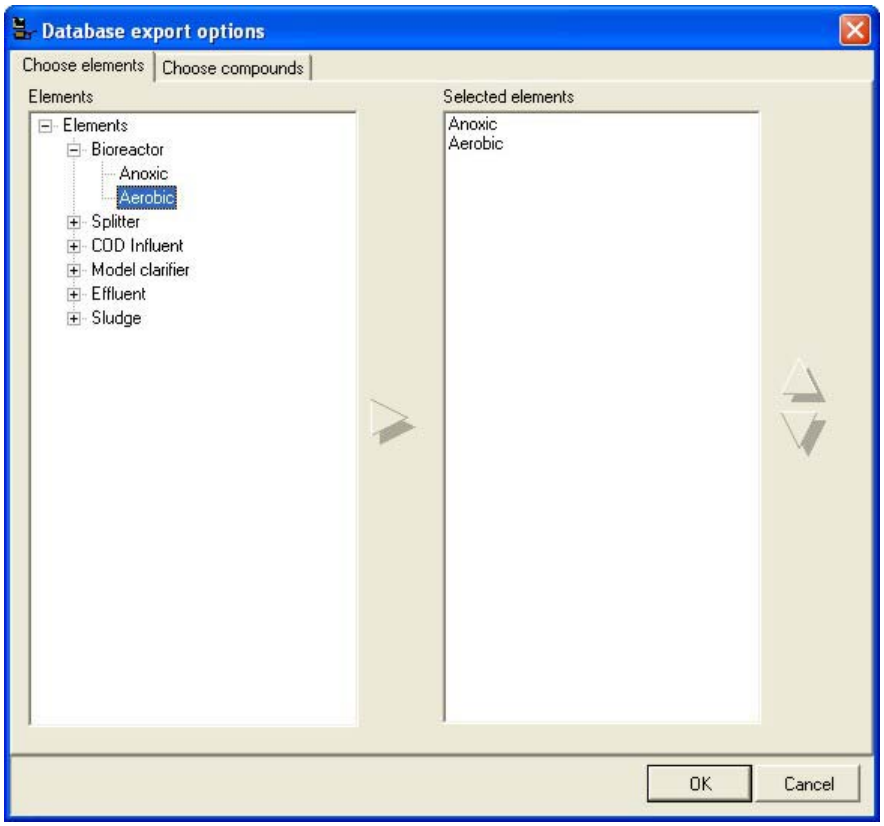
If you choose to export data to a file via the **Database|Custom export file…** command, you will be presented with the **Save File** dialog box. You may choose to export to one of the following file formats:

1. Text file (*.txt) – in this format, columns of data will be delimited by tabs, which is a suitable format for import to a spreadsheet application.

2. Comma-separated values (*.csv) – in this format, columns of data will be delimited by commas. This file format also may be imported to a spreadsheet application.
3. All file (*.*) – in this format, columns of data will be separated with a delimiter of your choice.

If you choose to export data to the clipboard via the **Database|Custom export clipboard…** command, no choice of delimiters is offered. Tabs will separate the data columns you paste to the clipboard since this is the best format for copying the data into a spreadsheet application.

Each of these commands allows you to select the elements that you want to export data for, using the following dialog box.



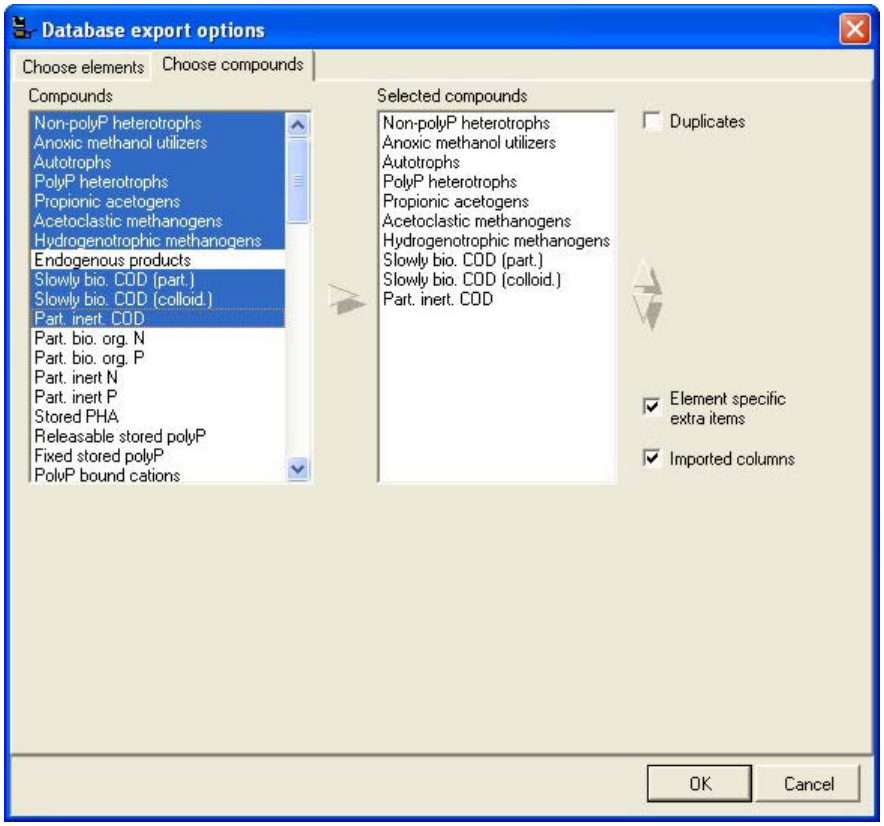
Dialog used to select elements for the custom export commands

From the **Elements** tree view, select the element(s) that you wish to export data for.

- If you wish to include all the elements from a group (for example, all the Bioreactors), click on the group heading and then click the right-pointing arrow to move them all to the **Selected elements** list.
- If you wish to include only certain elements from a group (or groups), then click on the plus sign (+) next to the group heading to expand it, click on the specific element you want to include, and then click the right-pointing arrow to move it to the **Selected elements** list.

- If you want to change the order of the **Selected elements** columns in the export, move the elements around by clicking on them and then clicking the Up/Down arrows. In the above example, the leftmost columns of your exported data would be occupied by monitored parameters you have exported for the element named **Unaerated**, followed by those for the **Aerobic** element.

Once you have selected the elements to export data for, click on the **Choose compounds** tab to select the parameters that you want to export for the selected elements. This tab is shown below:



Dialog used to select parameters for the custom export commands

In the **Compounds** list, select the compounds that you wish to appear in your table, and add them to the **Selected compounds** list by clicking the right-pointing arrow. To select multiple compounds:

- Select contiguous multiple compounds by clicking on the first desired compound, holding down the **Shift** key, and clicking the last desired compound.
- Select non-contiguous multiple series by holding the **Ctrl** key and clicking on the different desired compounds.

If you want to change the order of the **Selected compounds** in the export, move the compounds around by clicking on them and then clicking the Up/Down arrows. In the above example, the columns (from left to right) of your exported data would be occupied by monitored parameters **Zbh**, **Zba**, **Zbp**, **Zbpa**, **Xsp**, and **Xsc** for the

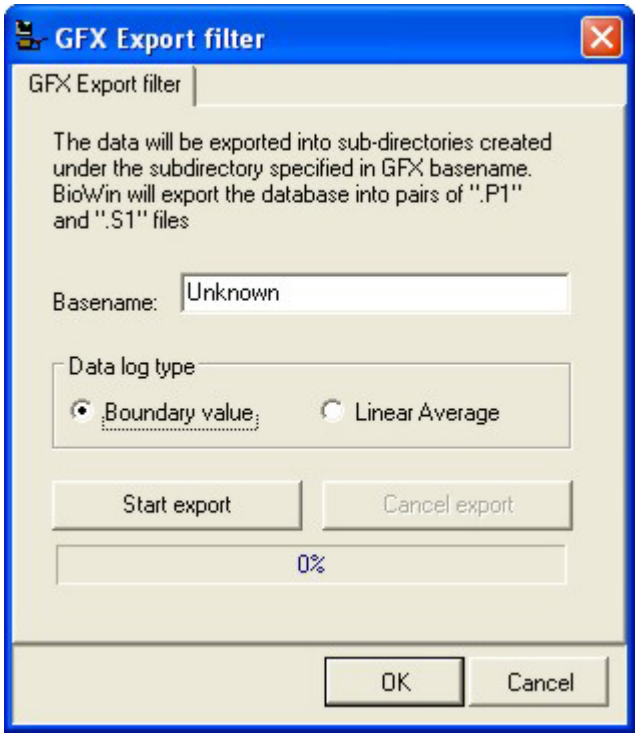
element named **Unaerated**, followed by **Zbh**, **Zba**, **Zbp**, **Zbpa**, **Xsp**, and **Xsc** for the **Aerobic** element.

You may have set up monitoring of some element-specific parameters such as solids loading rate for a secondary clarifier element, or OUR for a bioreactor element which you cannot select from the generic **Compounds** list. If you wish to include these parameters in your export, place a check in the box labeled **Element specific extra items**. By checking the box labeled **Imported columns**, you can also export any data that you have previously imported to the database.

Note : using these two dialogs to export data requires that you have previously set up monitoring for the parameters in the appropriate elements, and run a dynamic simulation. Use of these dialogs to export data does not set up monitoring.

Exporting To GFX Files

You can export the database in **GFX** format. When you execute this command, you will be presented with the following dialog box.



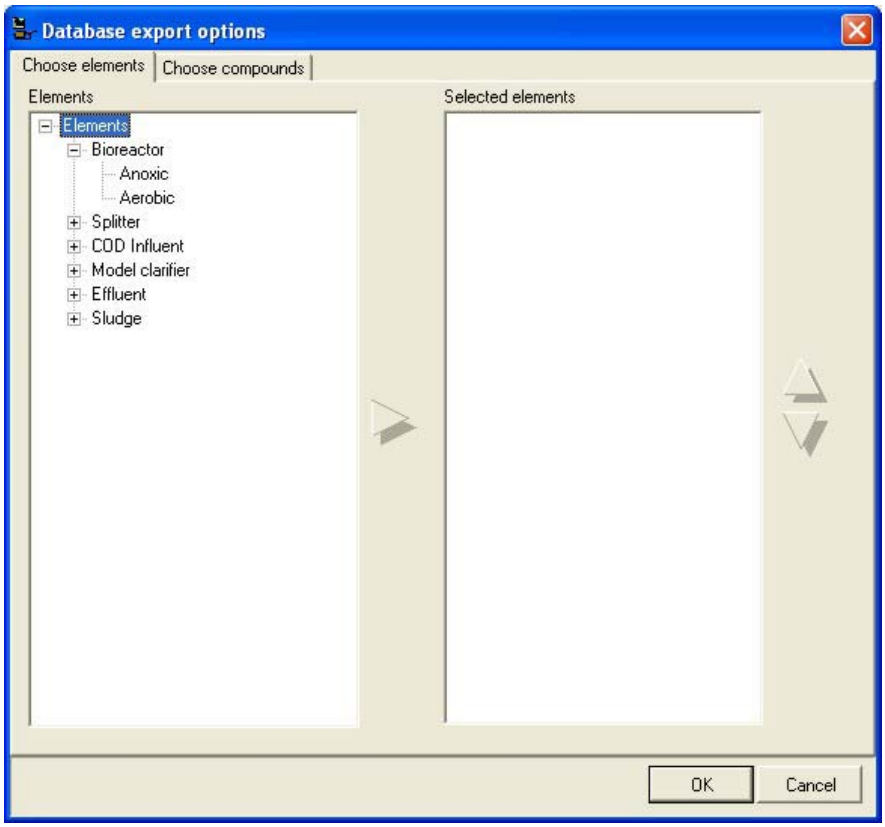
Dialog used for setting up database export to GFX file

GFX *.Pn files may contain up to 16 parameters (e.g. VSS, TSS, CODt, etc.). **GFX *.Sn** files contain the data corresponding to the ***.Pn files**. The data are in hexadecimal integer format, and missing parameters (i.e. if the **Pn** file contains less than 16 parameters) are padded with **FFFF**. These files will be stored in the subdirectory name that you specify in the **Basename** text entry area. This subdirectory will be located beneath the BioWin **Data** directory.

You must specify the **Data log type** as **Boundary value** or **Linear average**. **GFX** assumes 15-minute intervals; data logged to this file are expected to be the average for the 15 minutes (for the best approximation of this you can set the BioWin data interval to 15 minutes).

- A **Boundary value** export uses data taken from the BioWin database at 15-minute intervals.
- A **Linear average** export uses data taken from the BioWin database at 15-minute intervals, but the data are linearly interpolated at the midpoints (i.e. 7.5, 22.5, 37.5 minutes) using surrounding data.

Clicking the **Start export** button allows you to select the elements that you want to export data for, using the following dialog box:



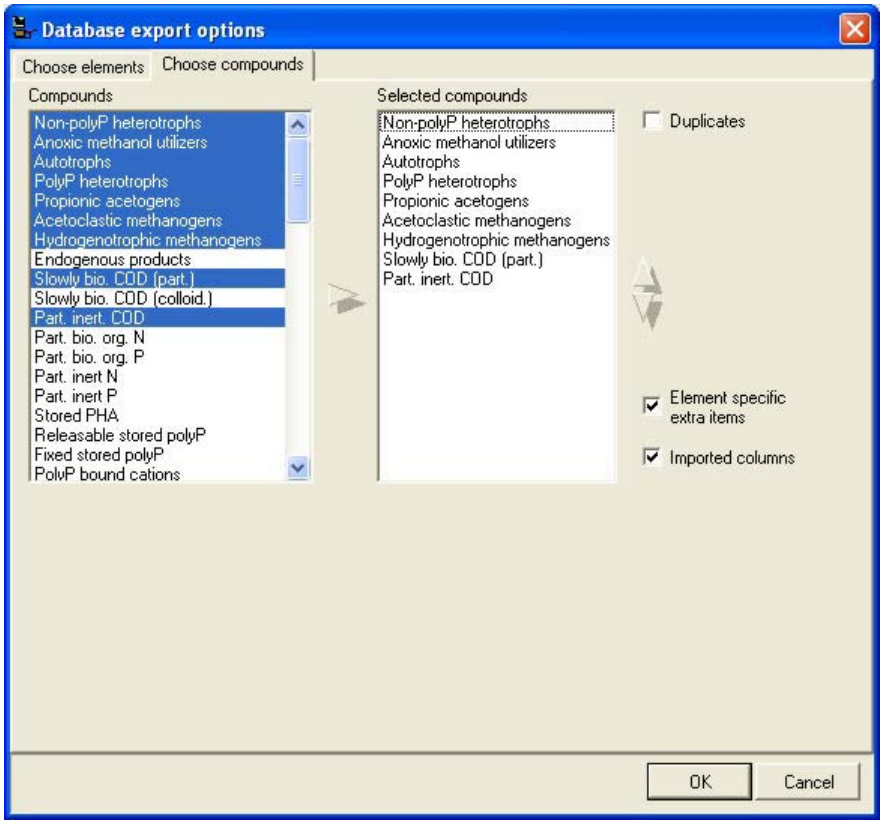
Dialog used to select elements for the GFX export

From the **Elements** tree view, select the element(s) that you wish to export data for.

- If you wish to include all the elements from a group (for example, all the Bioreactors), click on the group heading and then click the right-pointing arrow to move them all to the **Selected elements** list.
- If you wish to include only certain elements from a group (or groups), then click on the plus sign (+) next to the group heading to expand it, click on the specific element you want to include, and then click the right-pointing arrow to move it to the **Selected elements** list.
- If you want to change the order of the **Selected elements** columns in the export, move the elements around by clicking on them and then clicking the Up/Down arrows. In the above example, the leftmost columns of your exported data would be occupied by monitored parameters you have exported for the

element named **Unaerated**, followed by those for the **Aerobic** element.

Once you have selected the elements to export data for, click on the **Choose compounds** tab to select the parameters that you want to export for the selected elements. This tab is shown below:



Dialog used to select parameters for the GFX export

In the **Compounds** list, select the compounds that you wish to appear in your table, and add them to the **Selected compounds** list by clicking the right-pointing arrow. To select multiple compounds:

- Select contiguous multiple compounds by clicking on the first desired compound, holding down the **Shift** key, and clicking the last desired compound.
- Select non-contiguous multiple series by holding the **Ctrl** key and clicking on the different desired compounds.

If you want to change the order of the **Selected compounds** in the export, move the compounds around by clicking on them and then clicking the Up/Down arrows.

You may have set up monitoring of some element-specific parameters such as solids loading rate for a secondary clarifier element, or OUR for a bioreactor element which you cannot select from the generic **Compounds** list. If you wish to include these parameters in your export, place a check in the box labeled **Element specific extra items**.

Note : using these two dialogs to export data requires that you have previously set up monitoring for the parameters in the appropriate elements, and run a dynamic simulation. Use of these dialogs to export data does not set up monitoring.

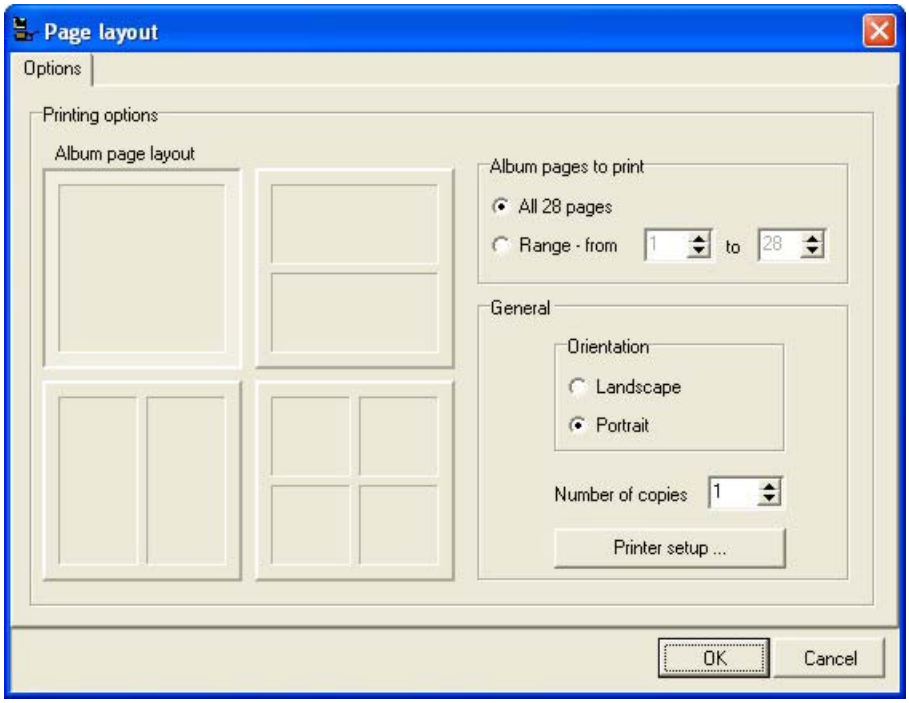
Once you have selected the elements and compounds for exporting to the **GFX** file, click **OK** to begin the export process. The status bar on the main exporting dialog will show the progress of the export process. When this is finished, you can find the files in the folder with your specified **Basename** under the BioWin **Data** directory.

Creating Project Reports

There are two choices for generating reports containing information about the currently opened BioWin configuration. The two options are **Report to printer**, and **Report to Word™**. Each option is explained in more detail below. The information that is included in reports may be specified using the **Report options** tab accessed via the **File|Report** menu command.

Report to Printer

This command will generate a printed BioWin report. When the command is selected, the following dialog box for controlling the layout of album pages in the report is opened.



Dialog box used for controlling printed report layout

The **Album pages to print** section allows you to choose between printing **All X pages** (X changes according to the number of pages in the current album), or a **Range** of pages (when the option is selected, you may enter text in the **from** and **to** spin edit boxes).

Once you have selected the album pages that you want to include in the report, you can choose between one of four **Album page layout** options:

5. The top left option will result in the contents of each album page being printed on an entire page in the report. For example if an album page contains only one chart, that chart will be printed on an entire page in the report. If an album page contains two panes with a chart on each, then the two charts will be printed on an entire page in the report.
6. The top right option will print the contents of each album page to one horizontal half of a page in the report. Each page of the printed report will contain two BioWin album pages.
7. The bottom left option will print the contents of each album page to one vertical half of a page in the report. Each page of the printed report will contain two BioWin album pages.
8. The bottom right option will print the contents of each album page to one quarter of a page in the report. Each page of the printed report will contain four BioWin album pages.

The **General** section contains a number of generic printing options. You can select your report **Orientation** to be **Landscape** or **Portrait**. You can specify the **Number of copies** of your report that you would like to print. The **Printer setup…** button will open the printer setup dialog box which will allow you to access the printer's properties, set paper size, page orientation, and a number of other printer options (the options presented to you will be dependant on the printer you have).

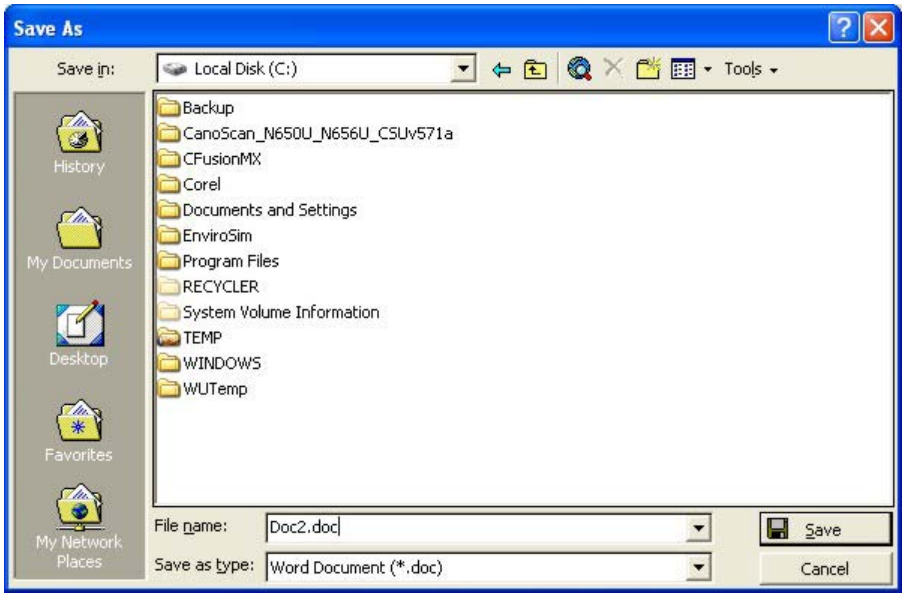
Once you are satisfied with your report layout settings, click the **OK** button to send the report to the printer.

Report to Word

This command will generate a report in Microsoft Word™ document format. Once you have generated the Word™ document, you may print it out, add your own text to it, copy sections of it into your company report documents, etc.

Once you have specified the information that you want to be included in the report using the **Report options** tab accessed via the **Tools|Customize** menu command, select the **File|Report|Report to Word™** command to begin the document generation process.

After a few seconds, the Microsoft Word™ **Save As** dialog box will open. An example of the dialog box is shown below (appearance may vary according to Windows system settings).



Word™ Save As dialog box

Give your report document a meaningful name, and make sure you note the location you save it to (you may navigate to any folder, just as you would saving any document).

Note : The report must be given a unique name. You may not "save over" an existing file.

Once you click the **Save** button, Word™ will open after a few seconds (time may vary depending on the speed of your computer). You will then see material being placed in Word™. During this time your cursor likely will take the Windows "hourglass" form.

Note : It is important not to attempt to perform any other operations while the report is being generated. BioWin and Word make heavy use of the Windows clipboard during this operation, so any attempts to copy / paste may cause problems in the report generation process.

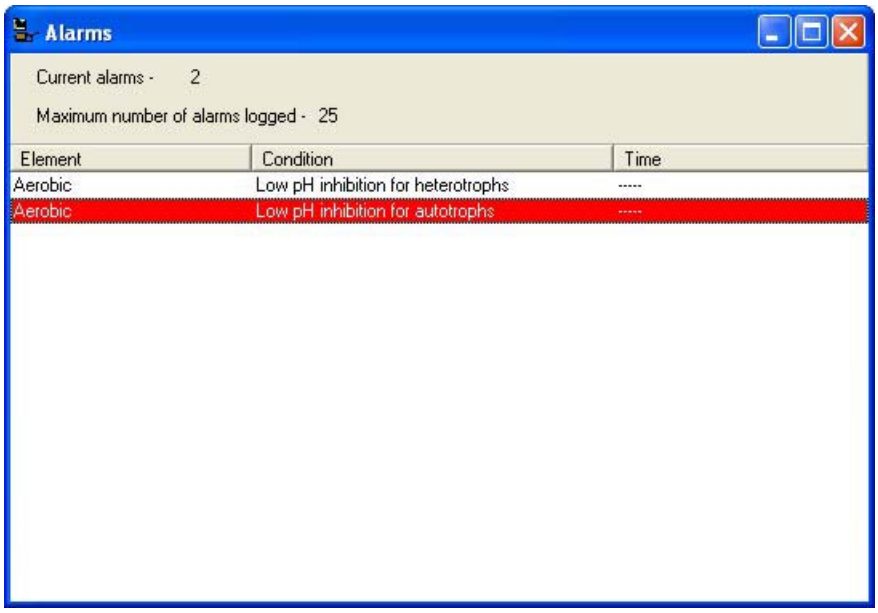
When the report is complete, Word™ will be closed and BioWin will return to the foreground. You may then locate the file and do what you wish with it.

Alarms

BioWin will notify you when certain conditions occur while you are performing a simulation. At the end of a steady state simulation, BioWin will always display the following alert if any alarm conditions have been detected:

Notification displayed at the end of a steady state simulation if alarm conditions have been detected

If you have set the **Alarm list** (on the **General** tab accessed via **Tools|Customize…** menu command) to any value greater than zero, clicking **OK** on the above notification will display a list showing the element in which the alarm condition occurred and the alarm condition. The list also can be viewed at any time via the **View|Alarms** command. An example of such a list is shown below.



An example of an alarm list

The alarm list also shows you the number of **Current alarms** (this information also is shown in the main window status bar), and the **Maximum number of alarms logged** (set in the **Alarm list** on the **General** tab accessed via **Tools|Customize…** menu command).

If you have **un-checked** the box labeled **Suppress alarms in dynamic simulations** (on the **General** tab accessed via the **Tools|Customize…** menu command), BioWin will log alarm conditions encountered during dynamic simulations. For these alarms, the time when they occurred is displayed in the alarm list (see third column in the screen shot above). The list will be displayed when you choose to end a dynamic simulation.

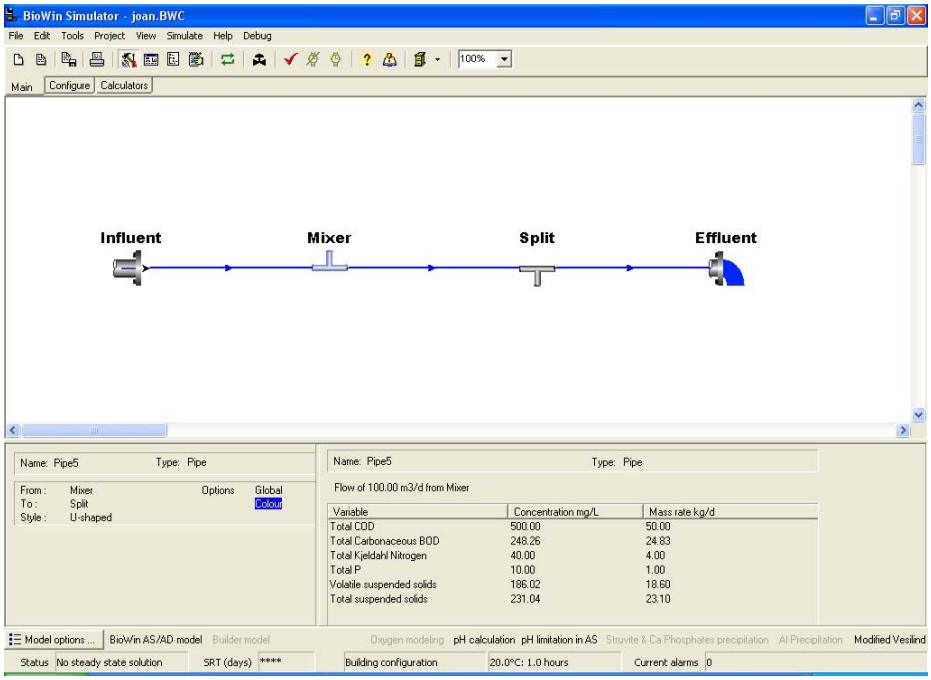
You may clear an alarm list by right-clicking in the list and selecting **Clear** from the resulting popup menu.

Alarm conditions

This section describes the various alarms that you may encounter in a project.

Recursive Configuration

You will be presented with this warning if your configuration is laid out in such a manner that you are attempting to connect a node splitter directly to a node mixer without first passing through an element that has a volume. The simplest case illustrating this is shown below.



A simple example of a recursive configuration

This situation is not allowed because there is no physical meaning to two node elements joined in sequence and the flow solver does not allow this physical impossibility.

Flow Specifications Could Not Be Achieved

This alarm occurs when the flow rate specified by the user could not be achieved because other constrains within the system override the user setting. For example; the flow rate specified will result in negative flows elsewhere in the system, a variable volume tank with a specified outflow rate has run dry (or is overflowing), a constant rate split specification exceeds the influent flow to the splitter.

Nitrogen Limited Conditions Occurred

This alarm occurs when the nitrogen (ammonia and nitrate nitrogen) in a biological zone is sufficiently low to cause a 50% or more reduction in the reaction rates.

Phosphorus Limited Conditions Occurred

This alarm occurs when the phosphorous (PO4) in a biological zone are sufficiently low to cause a 50% or more reduction in the growth reaction rates.

Magnesium Or Cation Limitation For P Uptake

This alarm occurs when the magnesium or cation concentrations in a biological zone are sufficiently low to cause a 50% or more reduction in the in the biological uptake of phosphorus by polyphosphate accumulating organisms.

More Than 10% Of Ammonia Is Stripped

This alarm occurs when it appears that 10% or more of the influent ammonia to a biological zone is being stripped by the gas phase.

The Air Supply Required To Achieve The DO Setpoint Exceeds The Maximum Air Supply Rate

This alarm occurs when the air supply rate required to achieve the DO setpoint entered by the user is higher that the maximum air supply rate specified by the user.

The Air Supply Requested Exceeds The Maximum Air Supply Rate

This alarm occurs when the air supply requested is higher that the maximum air supply rate (as specified by the user).

The Power Supply Required To Achieve The DO Setpoint Exceeds The Maximum Power Supply Rate

This alarm occurs when the power supply rate required to achieve the DO setpoint entered by the user is higher that the maximum power supply rate specified by the user.

The Power Supply Requested Exceeds The Maximum Power Supply Rate

This alarm occurs when the power supply requested is higher that the maximum power supply rate (as specified by the user).

High Air Flow / Diffuser

This alarm occurs when the air flowrate per diffuser exceeds the maximum air flowrate per diffuser specified by the user.

DO In Tank Is Higher Than Specified Setpoint Due To DO In Input (Even Without Aeration)

This alarm occurs when the DO setpoint specified by the user could not be achieved because the influent DO is higher than the DO setpoint and there is insufficient oxygen demand to lower the DO to the setpoint specified.

Low / High pH Inhibition

This alarm can apply to:

- Autotrophs
- Heterotrophs
- Digesters

These alarms occur when the pH in a biological zone is at a level where it is inhibiting the growth rate by 50% or more.

pH (or IS) Could Not Be Calculated

This alarm (including the High or Low bound varieties) occurs when there is an error calculating the pH or ionic strength.

Warning: High Ionic Strength (Activity Coefficients)

These alarms occur when the ionic strength is too high to calculate the approximate activity coefficients. The activity coefficients approximations calculated at an ionic strength of 3 are used instead.

Customizing BioWin

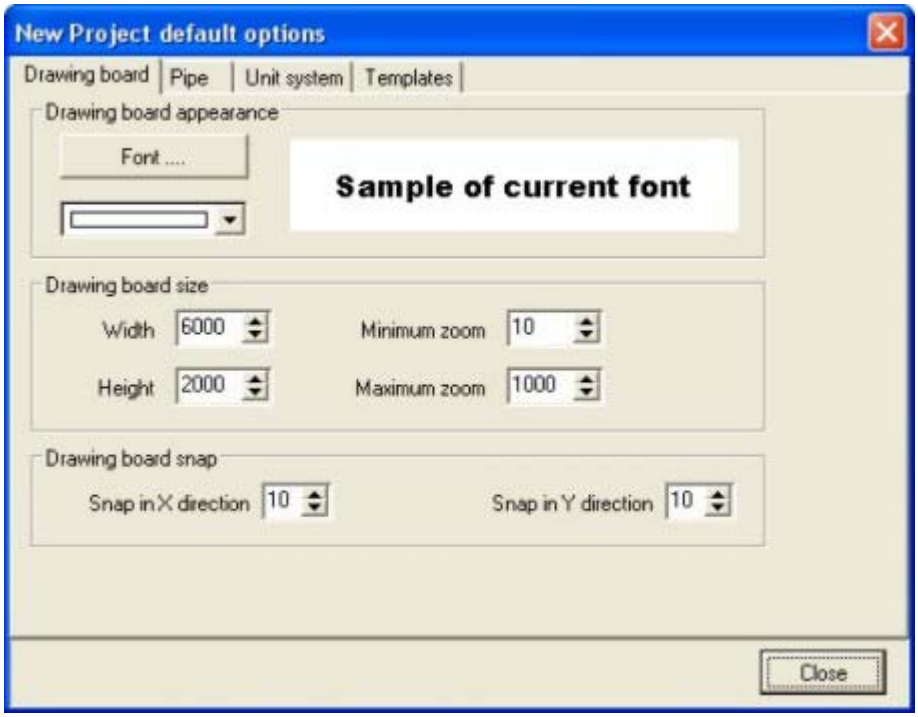
Customizing the Project Appearance

There are a number of parts and features of BioWin that can be customized to look how you want. When you customize BioWin, you essentially are changing the default settings of the BioWin work environment and all new projects that are created therein. The following sections discuss the dialog boxes that are accessed via the **Project|New Project Options** menu command that may be used to customize BioWin.

The difference between customizing BioWin and setting individual project options should be emphasized. For more information on the various project options which may be set for individual projects, refer to the "*Managing BioWin Projects*" chapter.

Drawing Board Options

This section outlines the various project options that can be set for the default drawing board. The drawing board options may be accessed in the main simulator window by choosing **Project|New Project Options** from the menu and selecting the **Drawing board** tab, shown in the picture below.



New project drawing board options tab

Aspects of the **Drawing board appearance** that may be changed include the **Font** and the **Color**. Clicking the **Font…** button will open the font properties dialog box and will allow you to change the font that is used to label element pictures on the drawing board. Selecting a new color from the drop list box will change the background color of the drawing board. Notice that a preview of the drawing board background color and the selected font is given in the dialog box.

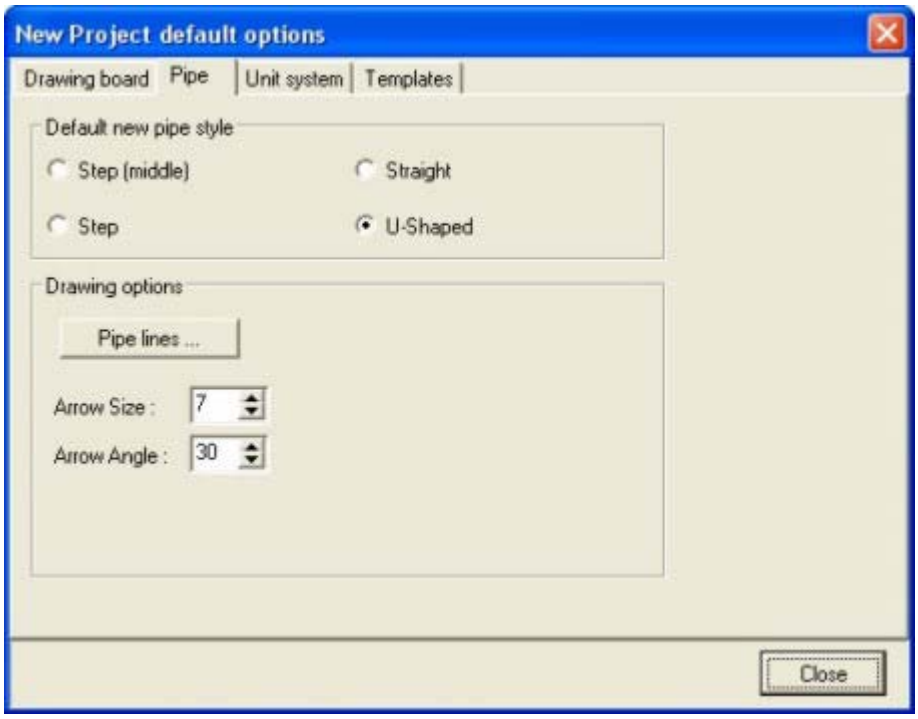
The **Drawing board size** also may be changed. Changing the values in the **Width** and **Height** spin edit boxes will change the overall dimensions of the drawing board. The size of the main window occupied by the drawing board is not changed, however, the overall size is changed as evidenced by a change in the size of the scroll box in the scroll bar. The default values of 6,000 by 2,000 translate roughly to 60 by 20 inches. It should be mentioned here that if you plan to be performing copy and paste actions with Microsoft Word, the size of drawing board that you can copy into Word is limited to 22 by 22 inches. The **zoom limits** of the drawing board can be set so that zooming can only take place between minimum and maximum levels.

The **Drawing board snap** resolution can be changed in either or both the X and Y directions. The snap feature helps you align elements precisely on the drawing board. When you place or move an element on the drawing board, it aligns itself (i.e. “snaps”) to the nearest grid point (grid points are invisible). Therefore, increasing the snap values results in a coarser grid for the elements to snap to which means that you have less control over their placement. Decreasing the snap values results in a finer grid for the elements to snap to which means that you have increased control over their placement.

Pipe Options

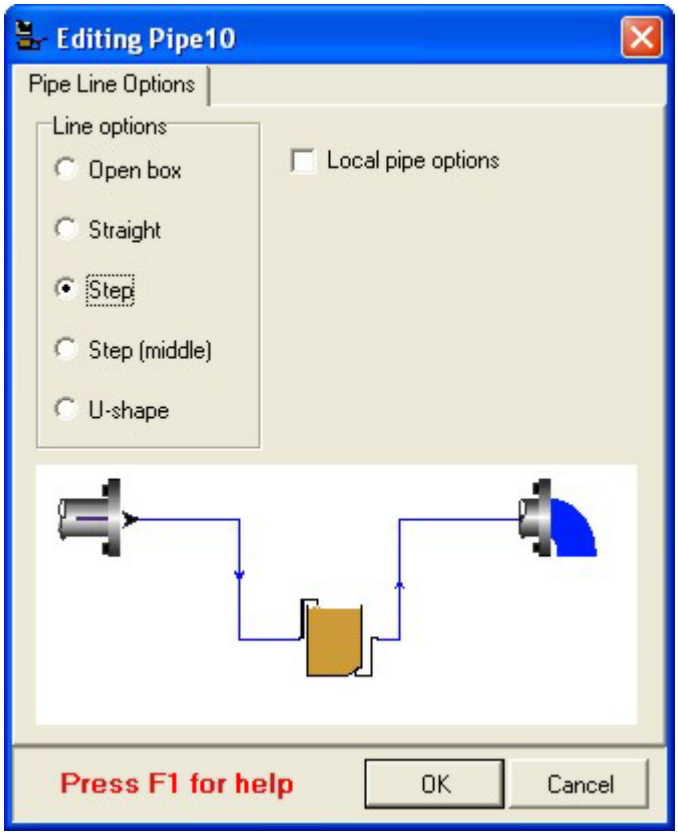
The "arrow angle" refers to the acute angle between the arrow side and the line. For example, if you want arrows with "flat" bases, set the arrow side angle to the maximum of 60 degrees.

You also can control the default appearance of pipes in your project, using the **Project|New Project Options** menu command and selecting the **Pipe** tab, shown in the figure below. To change the **Color**, **Width**, and **Style** of the lines used to represent pipes on the drawing board, click on the **Pipe Lines…** button. You may increase the **arrow size** on the lines used to represent pipes. The **arrow angle** also can be changed.



New project pipe options tab

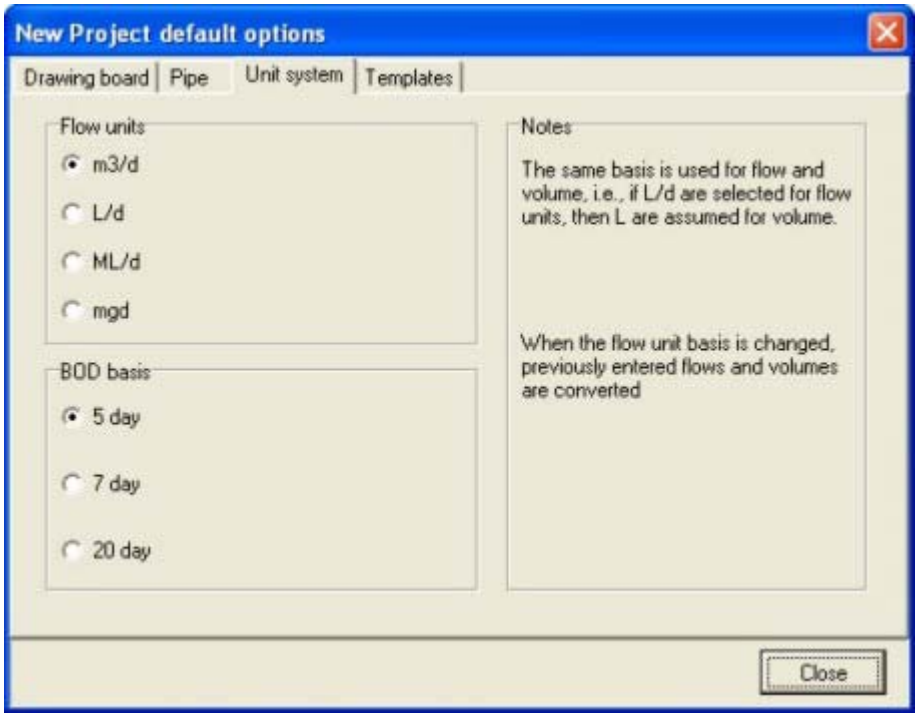
Note : It also is possible to change pipe line and arrow settings for individual pipes. Access the properties of the pipe you wish to change by double-clicking it or right-clicking it and selecting **Properties…** from the resulting pop-up menu. Next, check the box labeled **Local pipe options** and you will be presented with the same set of options that have just been outlined; however, these changes will affect this particular pipe only.



Pipe line options dialog box

Setting the Unit System

This section outlines the various options that can be set for the unit system used in a project. The project unit system may be accessed in the main simulator window by choosing **Project|New Project Options** from the menu and selecting the **Unit system** tab, shown in the picture below.



New project unit system options

In the **Flow units** radio button group, you may choose from the following:

- cubic meters per day **(m^3/d)**
- litres per day (**L/d**)
- megalitres per day (**ML/d**)
- megagallons per day (**mgd**)

It should be noted that the same basis is used for flow and volume. For example, if you choose **L/d** as your flow unit, then volumes (e.g. for bioreactors, settling tanks) will be in litres also.

Note : Air flowrates are in units of m^3/hr.

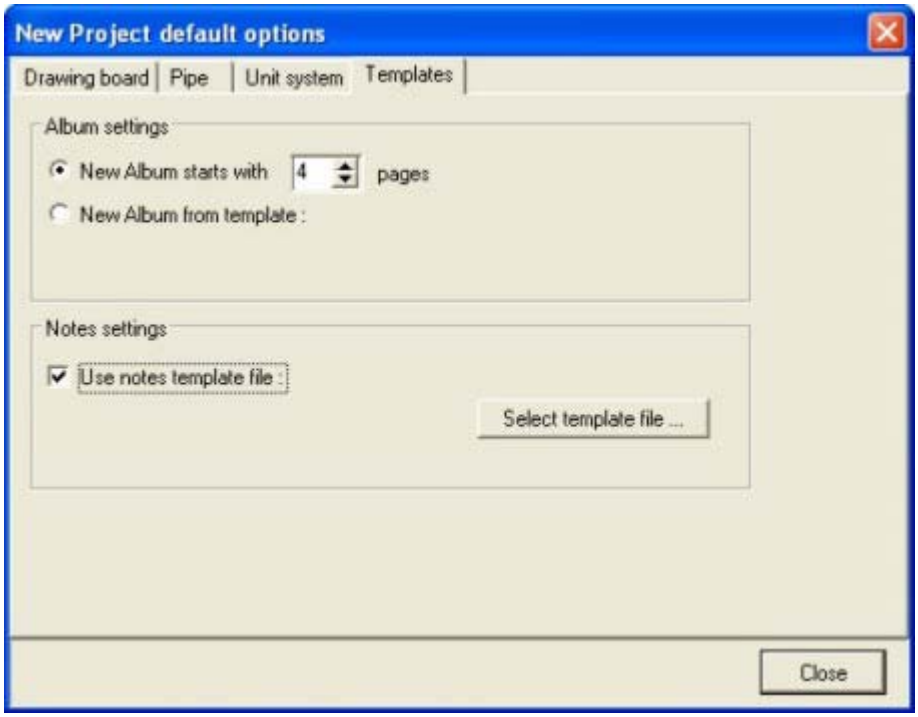
If you choose **mgd** as your flow basis, then the following imperial units will be used for other calculations in BioWin:

Measure	Imperial Unit
Length	feet (ft)
Area	square feet (ft^2)
Mass	pounds (lbs)
Pressure	pounds per square inch (psi)
Air Flow	Standard Cubic Feet per Minute
Specific Velocity	gal/ft^2/d
Mass Loading Rate	lbs/ft^2/d
Concentration	milligrams per litre (mg/L)

In the **BOD basis** radio button group, you may choose the length of time that BioWin uses to calculate BOD values. You may choose between **5**, **7**, and **20** day BOD.

Using Project Templates

This section outlines the various options that can be set for the album and notes templates you can use in a project. Templates for albums and notes may be accessed in the main simulator window by choosing **Project|New Project Options** from the menu and selecting the **Templates** tab, shown in the picture below.



New project templates dialog box

When you first open the BioWin album, it will contain a default number of pages. You can customize the number of pages using the **New Album starts with** spin edit box.

Note : These default pages contain one pane; if you want pages with multiple panes, you must add these manually.

If you have a number of album pages set up with pane and display configurations set to your liking, you can set this group of pages as your default album template file so that the album will open containing these pages. If you click the **New Album from template** radio button, the **Select template file…** button will be activated. Clicking this button will open a **Select template** dialog box allowing you to specify an album template pages (**.atp*) file you have previously created in BioWin.

The **Use notes template file** check box allows you to specify a file to use as a template when you execute the **Project|Notes** command. Checking this box will activate the **Select template file…** button which will open a **Select template** dialog box when it is clicked, allowing you to specify a **Rich Text Format** (*.RTF) or **Text** (*.TXT) file that you have created with another application such as Microsoft Word.

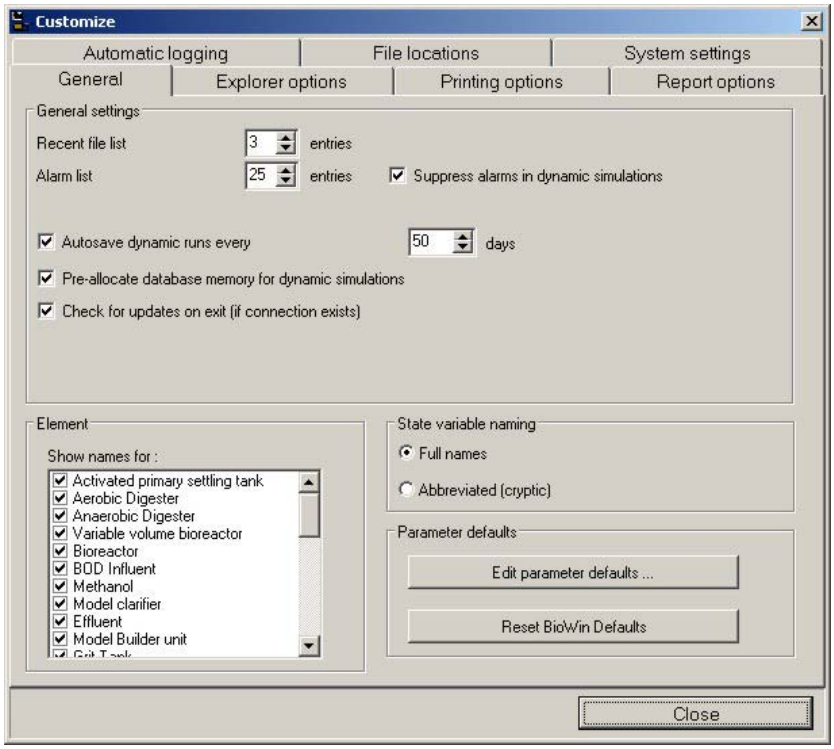
Customizing The Work Environment

There are a number of parts and features of BioWin that can be customized to behave how you want. When you customize BioWin, you essentially are changing the default settings of the BioWin work environment and all new projects that are created therein. The following sections discuss the dialog boxes that are accessed via the **Tools|Customize** menu command that may be used to customize BioWin.

The difference between customizing BioWin and setting individual project options should be emphasized. For more information on the various project options which may be set for individual projects, refer to the "*Managing BioWin Projects*" chapter.

General Options

The **General** tab, shown below, may be used to customize **General settings** for BioWin.



Tab used to customize general options

The number of files that are listed in the most recently used file list at the bottom of the main simulator window **File** menu can be changed using the **Recent file list** spin edit box.

The number of alarms that are displayed at one time in the alarm list can be controlled by using the **Alarm list** spin edit box. If you want BioWin to ignore alarms during dynamic simulations, place a check in the box labeled **Suppress alarms in dynamic simulations**. Checking this box stops BioWin from checking **any** alarm conditions, so no alarms will be logged during dynamic simulations regardless of the number you have entered in the **Alarm list** spin edit box.

You can select whether or not you want to be presented with the option for storing **previous time series plots in the album**. This option is useful for comparing

the results of one simulation run with others. When you restart a dynamic simulation, any series plotted from previous runs will remain on their respective charts. The saved series keep their names but will have a number appended to them indicating with which run they are associated. The naming convention is as follows:

- Saved series are numbered 0 to *n*-1
- Active series are numbered *n*

where *n* is the current simulation run number.

Note that you are able to format saved series as you would any other. This includes assigning them new names if you wish.

It is important to remember that saved series are simply lines on the charts for comparison purposes. They are no longer contained in the database. Perhaps this is best illustrated with an example. Say you conduct a dynamic simulation for five days and plot effluent ammonia. To assess the impact of the estimated maximum nitrifier growth rate, you change this model parameter. You start another five day dynamic simulation from the same start date as the first run, and save the effluent ammonia time series.

On your chart, there would be two series, "YourSeriesName#0" and "YourSeriesName#1". The series with 0 appended to its name is the saved series containing results from the first five day run. The series with 1 appended to its name contains results from the second five day run. Now suppose at the end of the second five day run you decide to continue for another 5 days. On your chart you would see "YourSeriesName#1" continuing to ten days, while "YourSeriesName#0" would be stopped at five days. If you exported the database at the end of the second run, you would export ten days of effluent ammonia data generated with the second maximum nitrifier growth rate.

You can specify whether you want BioWin to **save dynamic runs automatically**, and the frequency (in terms of dynamic simulation days) at which to save. If this option is selected, BioWin will create backup files named *filename*.X, where *filename* is the name you have selected for the currently open BioWin file. X will be integers indicating the number of automatic saves that have been performed. For example, say you set up BioWin to save dynamic runs automatically every 10 days, and then commence a 30 day dynamic simulation on a file you have saved as ***mybiowinfile.bwc***. At the 10th day of the dynamic simulation, BioWin will save a file called ***mybiowinfile.1***. At the 20th day of the dynamic simulation, BioWin will save a file called ***mybiowinfile.2***. At the 30th day of the dynamic simulation, BioWin will save a file called ***mybiowinfile.3***.

Most users will want to allow BioWin to **pre-allocate database memory for dynamic simulations**. When this option is selected, BioWin obtains the required amount of memory it will need before starting a dynamic simulation (the amount of memory required depends on the number of parameters being monitored in the database, the database monitoring frequency, and the length of the dynamic simulation). Selecting this option may result in a slight delay at the beginning of long dynamic simulations, but will avoid problems with BioWin slowing down during long dynamic runs.

A number of new charting features have been added to BioWin. If you prefer to use the charting interface of the previous version, you can select the option for BioWin to display **Simplified charting dialogs**.

The **Element** section may be used to customize the appearance of elements on the drawing board. You can choose the types of elements for which you want to **Show element names for** – by placing a check beside the desired elements.

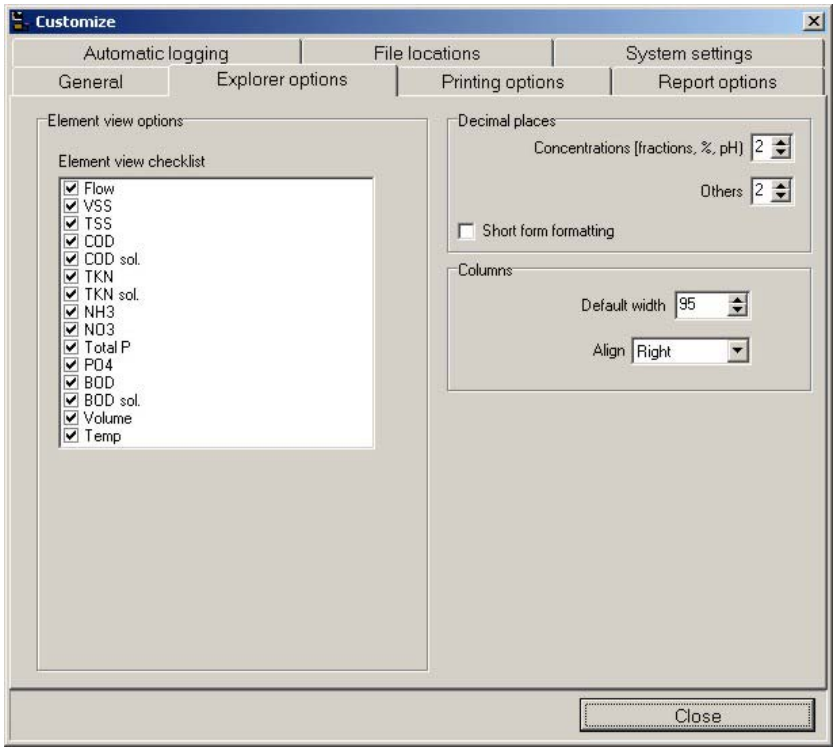
To control the display of state variable naming, use the radio buttons in the **State variable naming** section to choose between **Full names** and **Abbreviated**. For

example, the full name for nitrifying bacteria is “Autotrophs”, and the abbreviated name is “Zba”.

The default model parameters for BioWin can be controlled using the **Edit parameter defaults…** button in the **Parameter defaults** section. This button opens the **Parameter editor** dialog box (see **Model Parameter Editors** in the "*Useful BioWin Interface Tools and Techniques*" chapter). Parameters can be modified and printed from this location. Use the **Reset BioWin Defaults** button to restore the original settings for these parameters.

Explorer Options

Using the **Tools|Customize** menu command, using the **Explorer options** tab shown below, it is possible to customize the amount of information that is displayed in the right explorer pane when an element-type node or **State variables** node is clicked.



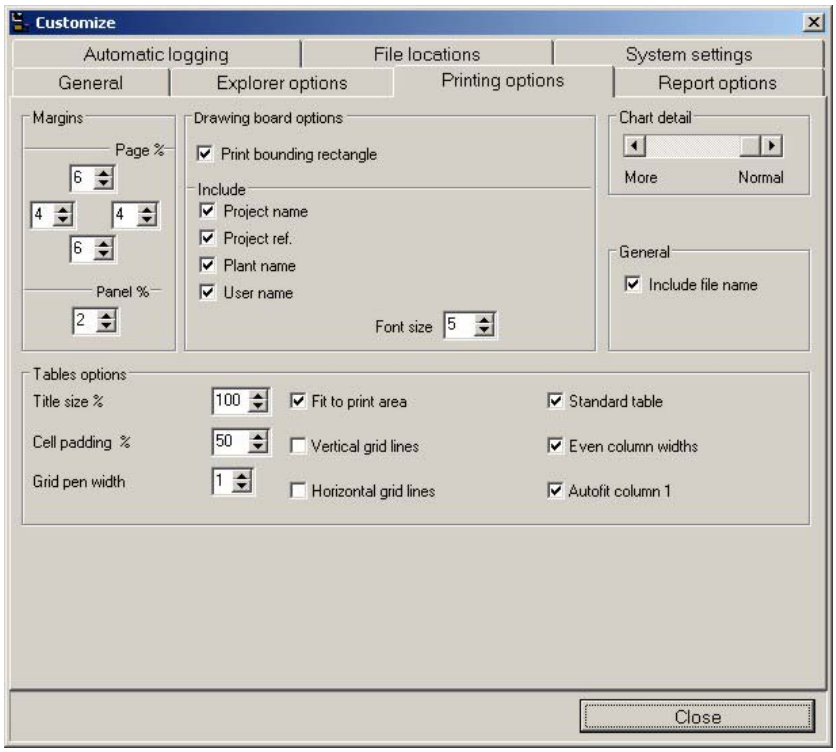
Tab used to customize explorer options

In the **Element view checklist**, you may specify the variables that will be displayed for an element when you click on its node or its parent node.

You can control how many decimal places are displayed in the Explorer window using the **Decimal places** spin edit box. You are able to have separate decimal place settings for **Concentrations** (e.g. ammonia concentration) and **Others** (e.g. flow values). You also may control the **Default width** of the columns used to display results in the Explorer, and the alignment of the data using the **Align** drop list box.

Print Options

The **Printing options** tab, shown below, may be used to customize print settings for BioWin.



Customize printing options dialog box

You can set up default printing **Margins**. With the **Page %** spin edit boxes in the section labeled **Margins**, you can set the top, bottom, left, and right margins to a desired set of values. Use the **Panel %** spin edit box to increase or decrease the white space between panes when you are printing out an album page.

There are some defaults that apply only to **Drawing board prints**. If you do not wish to print the border of the drawing board, then de-select the box labeled **Print bounding rectangle**. By checking or un-checking the boxes in the **Include** group, you can control whether or not the details entered via the **Project|Info…** command will display on the printout. If you choose to have some or all of this information displayed on drawing board printouts, then you can control the **Font size** using the spin edit box.

You can specify a default level of **Chart Detail** to adjust the text size on your print jobs. Sliding the scroll towards **More** decreases the size of text on your chart and gives greater prevalence to the chart on the printout. Sliding the scroll towards **Normal** increases the size of the text on your chart and gives less prevalence to the chart on the printout.

Note : When you are printing the drawing board or an album display, you will have the opportunity to override these defaults before finally sending the job the printer.

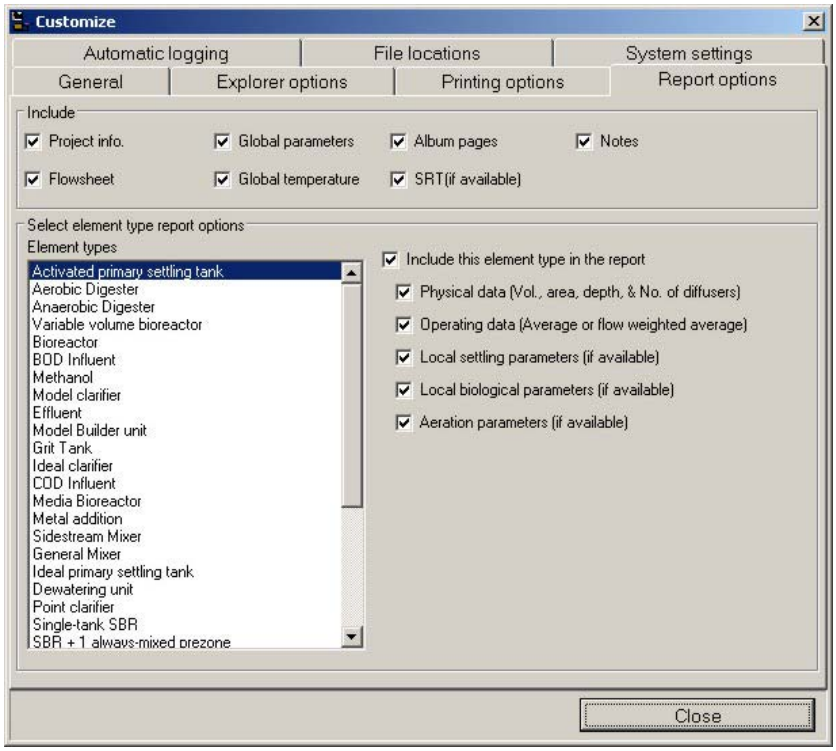
There also are a number of options that apply only to printing of **Tables**. The **Title size %** sets the size of the title font relative to the font size of the table text. Adjusting the **Cell padding %** changes the amount of space between the text in a table cell and the table cell boundaries. The **Grid pen width** controls the thickness

of grid lines drawn in the table. If you want the printed table scaled to fit into the print area that you have specified, select the **Fit to print area** option. You can control the drawing of grid lines by selecting or de-selecting the **Horizontal grid lines** and **Vertical grid lines** options. You also may choose to print your table as a BioWin **Standard table**, which is an elegant table with three horizontal lines: at the top of the table, underlining the title, and at the bottom of the table. To have equally sized columns in the table, select the **Even column widths** option. Since the entries in the first column quite often are longer than other columns (since they contain names), there is an option to **Autofit column 1** to avoid the truncation of the first column entries.

If you want to include the project file name on the drawing board printout or on table printouts, check the box labeled **Include file name**.

Report Options

You may customize the information that is included in reports generated by BioWin using the **Report options** tab, shown below.



Tab used to control BioWin report information

This tab may be used to specify what will be included in BioWin reports (both printed and Word™). The top **Include** section of the tab can be used to specify whether or not general information items will be included in the report. These are explained in the table below.

Item	Description
Project info.	Selecting this item will include information such as the Project Name, Plant Name, User name, file creation date, etc.
Flowsheet	Selecting this item will include a picture of the BioWin flowsheet in the report.
Global parameters	Selecting this item will include a tabular printout of all the

	model parameters.
Global temperature	Selecting this item will include the global temperature in the report.
Album pages	Selecting this item will include each album page (both tabular and chart).
SRT (if available)	Selecting this item will include the SRT in the report, if one is available.
Notes	Selecting this item will include the contents of the Notes editor in the report.

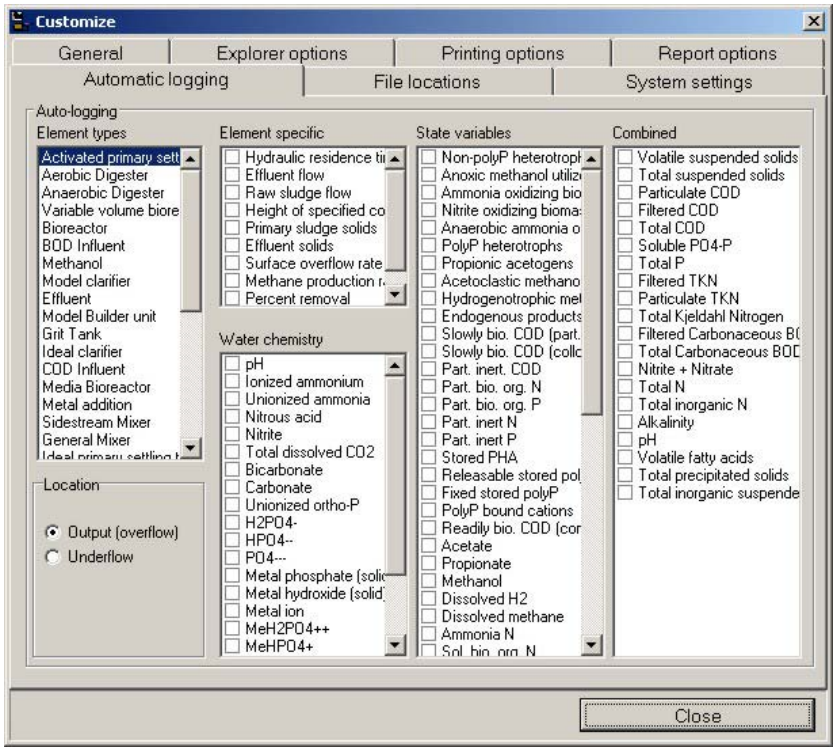
The contents of reports may be further customized on an element-by-element basis, using the **Select element type report options** section. The **Element types** list contains all the possible element types that could be included in a BioWin configuration. To include information for various element types in reports:

1. Click on the element type you want to include in the **Element types** list, so that it is highlighted blue.
2. Click the box labeled **Include this element type in the report**.
3. Click the boxes next to the different types of information (**Physical data**, **Operating data**, **Local settling parameters**, **Local biological parameters**, **Aeration parameters**).
4. Repeat this process for all the element types you wish to include in the report.

Note : Possibly not all of the types of information may apply to a given element – if so, it will simply not be included in the report.

Automatic Logging Options

You can customize BioWin to automatically monitor element data using the **Automatic logging** tab, shown below. For more information on managing data in BioWin, please see the **Managing Data** section.



Tab used to customize automatic logging settings

From the **Element types** list, select the element that you wish to set up auto-logging for. Using the **Location** radio button group, specify where you want to obtain element data from, i.e. **Input**, **Output**, or **Underflow**.

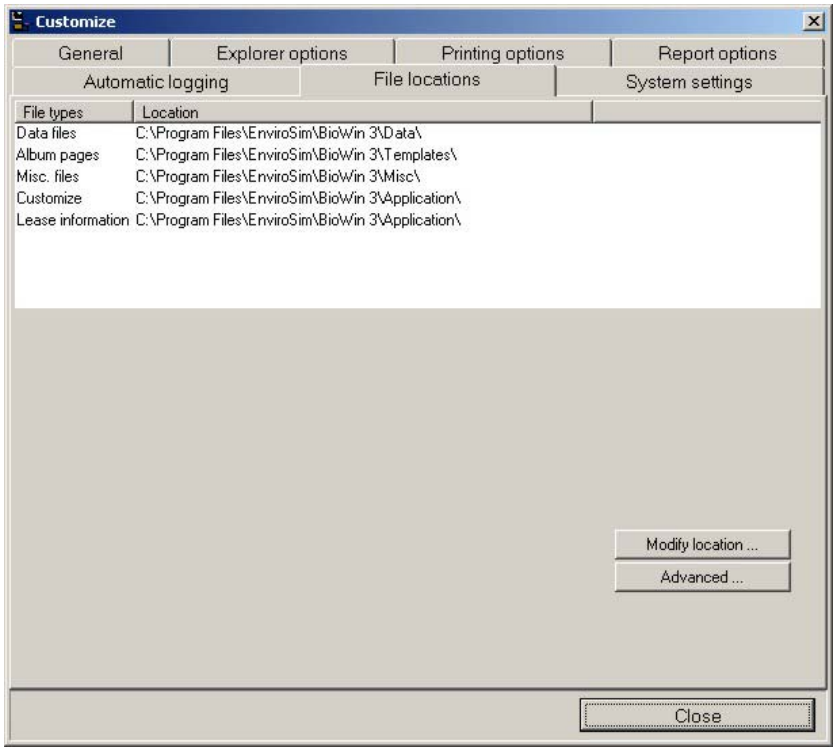
Note : The third location only will be shown for element types that have an underflow such as settling tanks.

Finally, in the **State variables**, **Combined**, **Water chemistry** and **Element specific** check list boxes; select the items that you wish to be logged automatically to the database.

When you set up auto-logging for an element type, the data that you specified to be monitored will automatically be written to the database for any elements added from that time forward, i.e. in the current project and all future projects (you can always modify the auto-logging options if you wish). Elements already in the configuration (or other configurations) are not affected by this setting

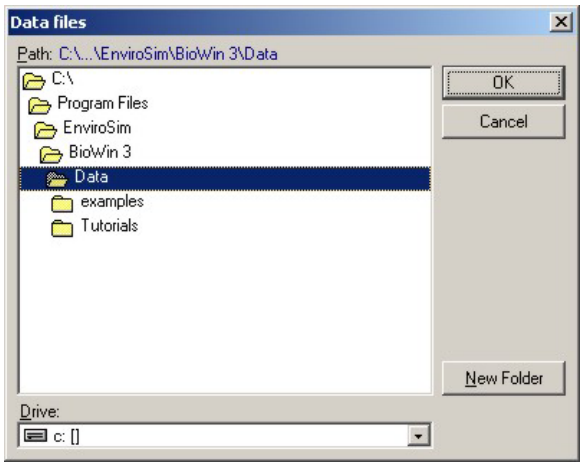
File Location Options

You may customize the locations where BioWin saves various file types using the **File locations** tab, shown below.



Tab used to customize BioWin file locations

To change the location of a file type, double-click on the file type's row of the file type/location list or click the row once and then click the **Modify location…** button. You will then be presented with a dialog box that you can use to specify a new location, such as the one shown below.



Dialog box used to specify new file locations

Select the drive you want to store the files on using the **Drive:** drop list box. Navigate through the existing folder structure by double-clicking on the folder icons. If you wish to create a folder, click the **New Folder** button. When you are satisfied with the new location, click **OK** to return to the **File locations** tab.

Clicking the **Advanced…** button will allow you to access the **Advanced file locations** tab. From this tab you may choose to specify folders where you have placed custom drawing board image files or BioWin binary files. ***Do not change***

these unless you are aware of the implications – this option is for advanced users only.

Chart Template Options

BioWin offers a wide range of formatting options so that you may customize charts. While this range gives users a great deal of power and flexibility in specifying the appearance of their charts, it also could result in a good deal of repetition if you plan to generate a number of charts which will have the same appearance. However, this repetition is eliminated by another useful feature of BioWin's charting package – chart templates.

The idea behind chart templates is to provide users with a means to pre-format their charts. You can format the chart template to have the appearance that you want the charts to have. Your current chart template (the chart shown in the **Chart Master**) is used as the basis for each new chart that you add to the album. To **change** existing charts you can **"Apply"** the current chart template to those charts. (See **Applying a Chart Template**).

Creating a Chart Template

To create a new chart template, select the **Tools|Chart Master** menu command to open the dialog box shown below.

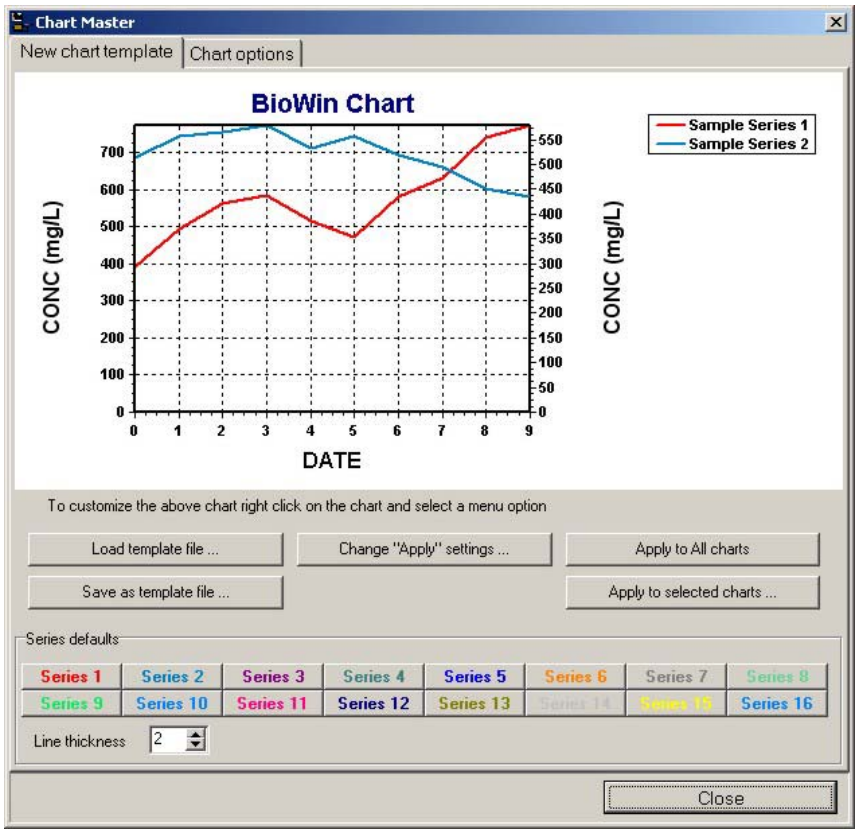


Chart Master used to customize the appearance of new charts

The default color and line thickness for each series can be edited using the **Series** (1-16) buttons and **Line thickness** spin edit box in the Series defaults section.

All other formatting options can be adjusted by double-clicking on the sample chart. This opens the chart editor dialog box as shown below.

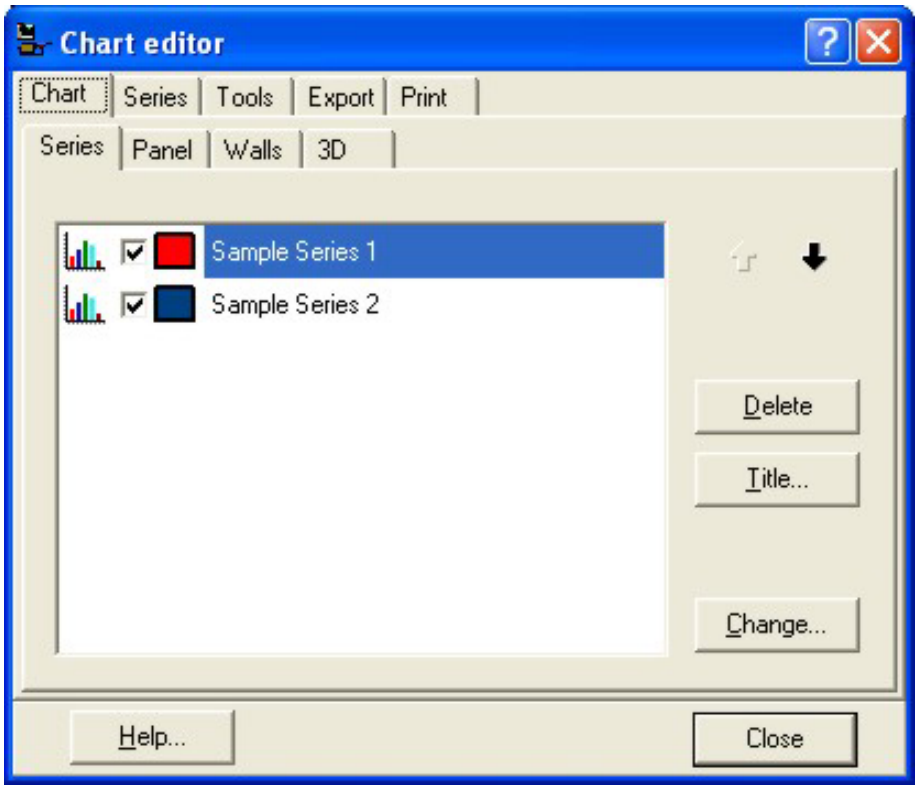


Chart editor dialog box

This dialog box allows you to adjust all other content and formatting aspects of the chart that you can incorporate into your template. Please refer to the *Chart Formatting Procedures* chapter for further information about using this dialog.

Once you have adjusted the settings as required, click the **Close** button to finish.

Saving a Chart Template

To create a new chart template, select the **Tools|Chart Master** menu command to open the dialog box shown below.

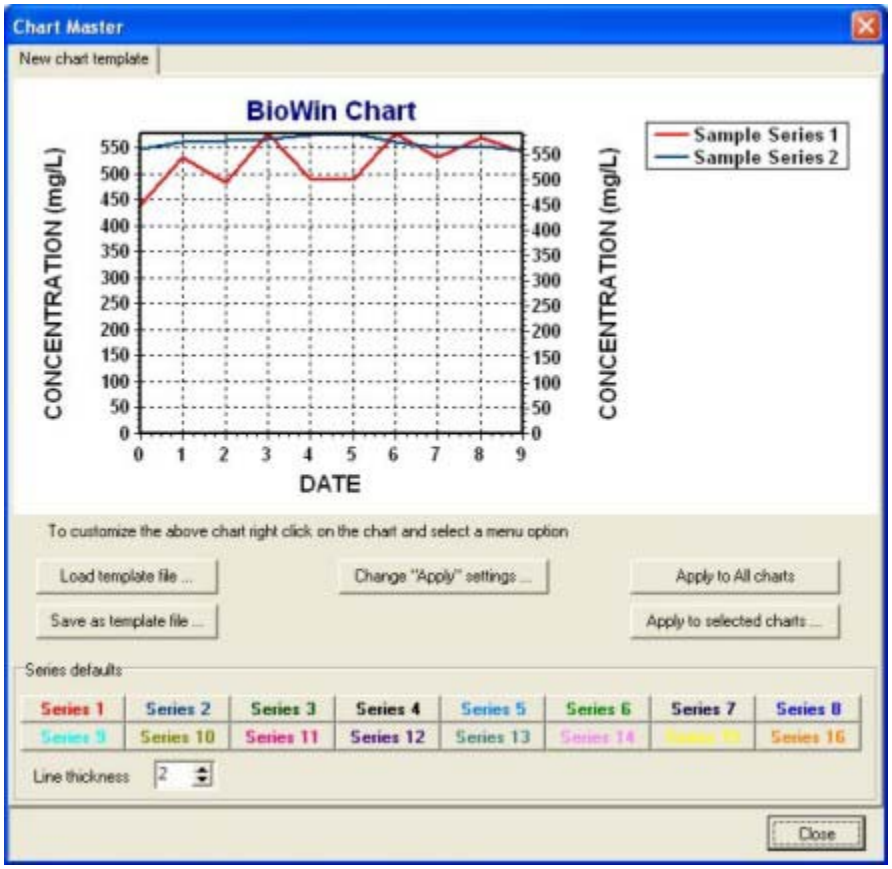


Chart Master used to customize the appearance of new charts

To save the chart formatting options you have set up into a template file for later use, click the **Save as template file…** button to open the **Save** dialog box. Chart template files are saved with a .BCM file extension. This file may be utilized later by clicking the **Load template file…** button and selecting the file from the **Open** dialog box.

Note: You may still customize the chart formatting using the normal series editing procedures in the album.

Applying a Chart Template

Once the desired template is loaded using the **Load template file…** button, you have the option of applying it all charts in the project or just selected charts. BioWin also allows you to control which specific formatting options will be applied from the template you are using.

1. To control the specific formatting options you want to apply from the template you have loaded, click the **Change "Apply" settings…** button to open the chart master options dialog box.

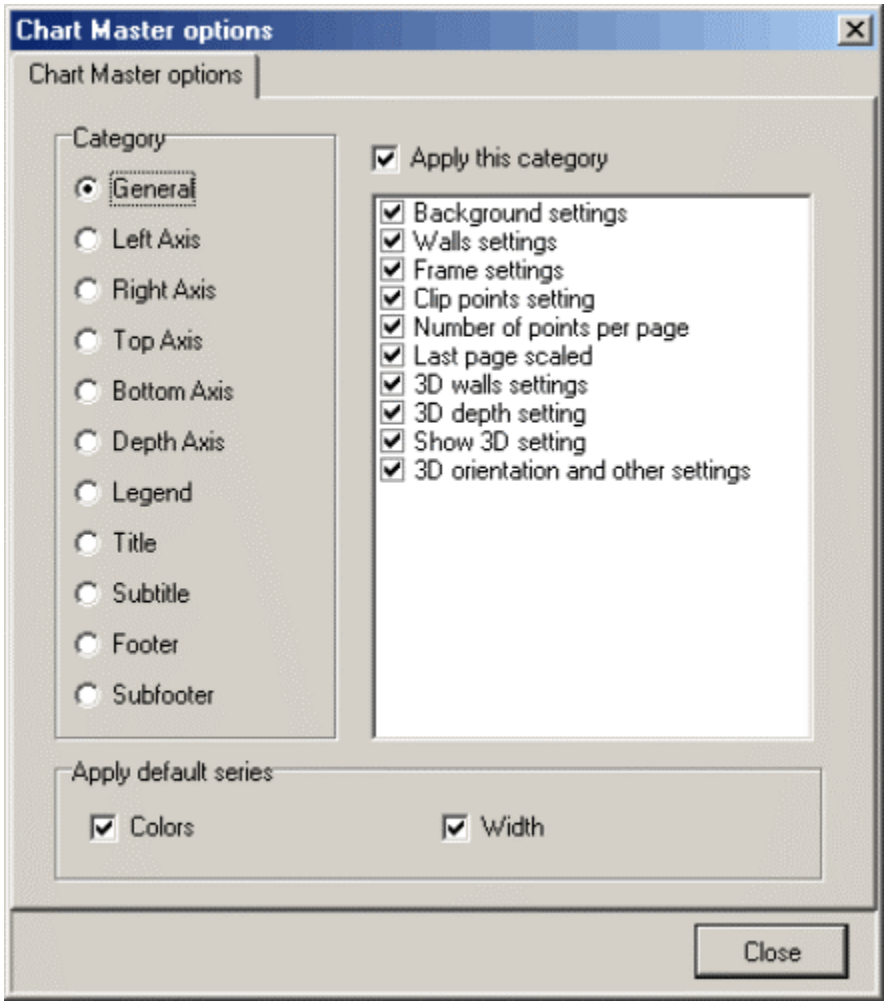


Chart master options dialog box

2. To apply a template to all charts in the project, click the **Apply to All charts** button.
3. To apply a template to selected charts only, click the **Apply to selected charts…** button to open the Selected charts dialog box, shown below.

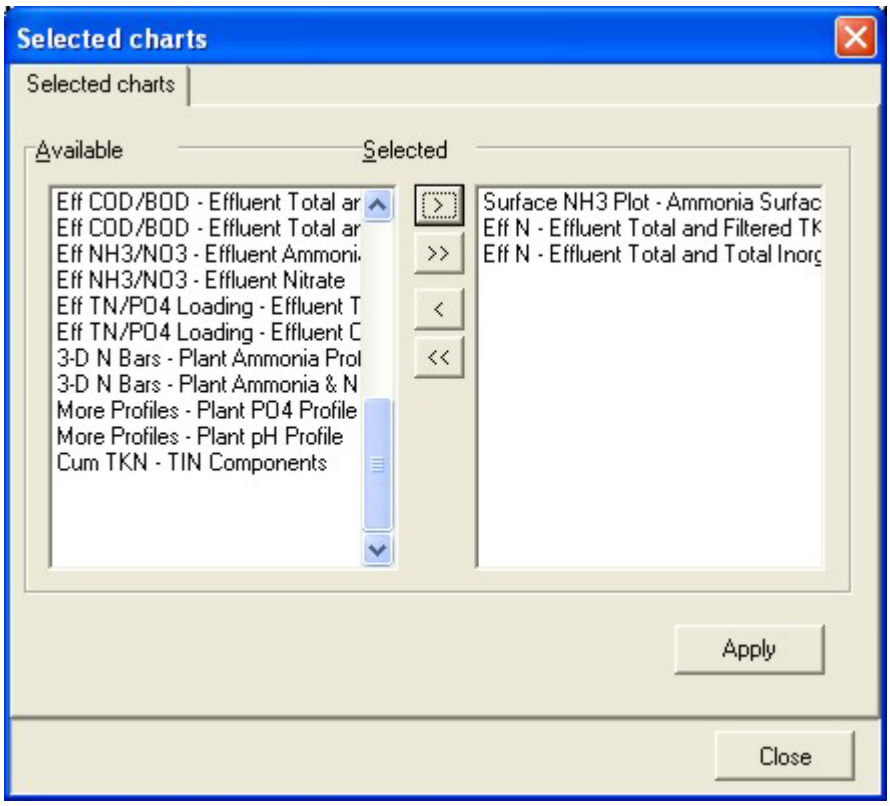
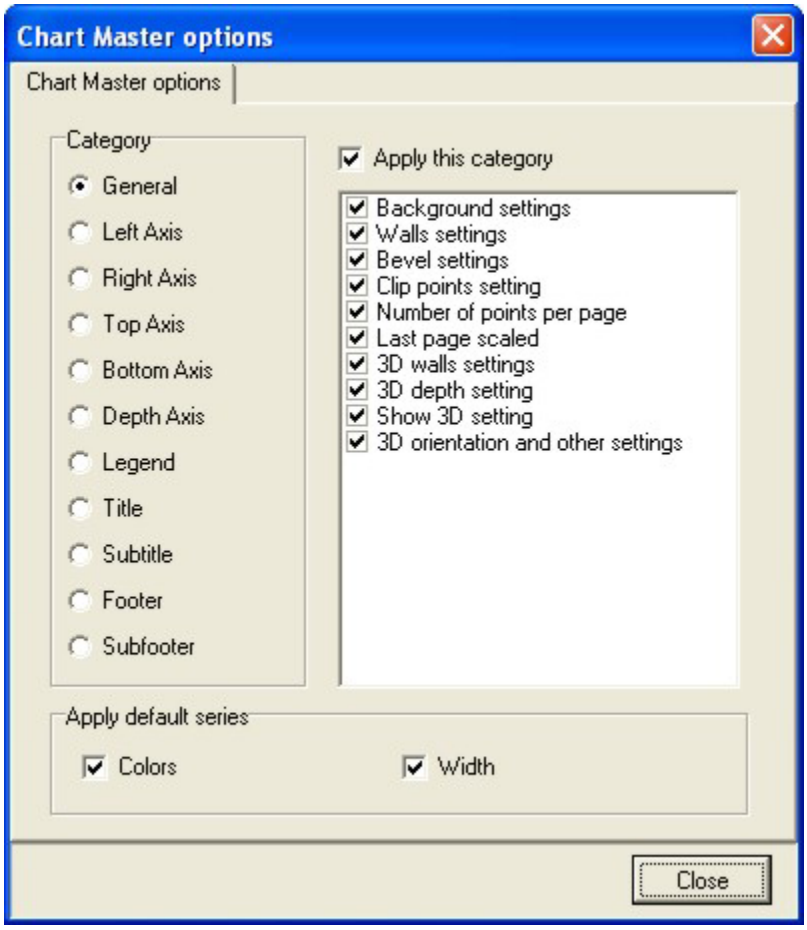


Chart selection dialog box

4. Click on the desired chart in the area labeled **Available** on the left and press the > button to add it to the **Selected** list on the right.
5. Click the **Apply** button, and then click the **Close** button to finish. The loaded template will then be applied to the selected charts only.

Changing Chart Master Options

The Chart Master allows you to control exactly which aspects of your template you wish to apply. For example, you may have already adjusted the Font used for your chart title, and do not wish to apply the Font used in your template. Chart Master will allow you to exclude this when you apply the template to your chart if you click the **Change "Apply" settings…** button.



Change apply settings dialog box

Each category will offer a variety of relevant settings with check boxes. By removing the check from a setting, you are removing that feature from the template.

Entire categories can be removed using the check box labeled **Apply this category**. If you uncheck this box, all options for the category will be ignored when the template is applied.

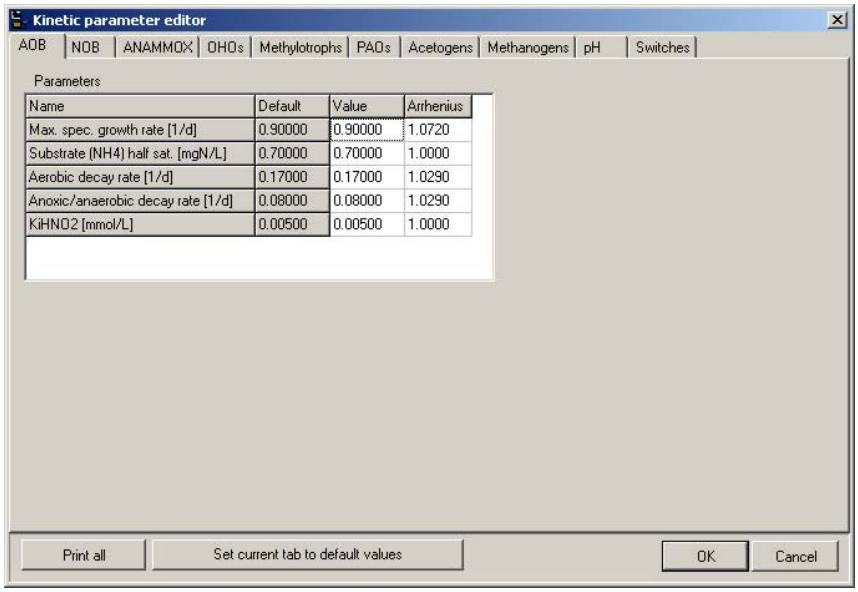
This dialog box also allows you to remove the **Colors** and **Width** settings in the section labeled **Apply default series**.

Useful BioWin Interface Tools and Techniques

Model Parameter Editors

BioWin model parameter editors, such as the example shown below, are used to change the parameters used in all of the various models employed by BioWin to perform simulations. Parameter editors are accessed via the **Project|Parameters** menu command.

Note : This section details the use of the model parameter editors. For technical information about model parameters, please see the "*Process Model Formulation*" chapter.



An example model parameter editor

Changing temperature dependency coefficients will increase or decrease the impact that a change in temperature will have on the model parameters.

In model parameter editors, a spreadsheet-like interface is provided for entering parameter values. Parameter names are listed in the first column, default values in the second column, and current values in the third column. In the case of kinetic parameters, temperature dependency coefficients for each parameter also are listed. Only the non-grey column may be modified. If you change a model parameter from the default value and accept your change, the value will be highlighted in bold red text the next time you view the editor.

To change a value, click on the cell you would like to modify and enter a new value, or click on the number in the cell to edit that value. You can use the arrow keys to move from one cell to the next. A number of options for copying and pasting data, printing the editor tab, and multiplying column values are available by right clicking on the editor tab - these are outlined below.

When you are working in a model parameter editor, a number of options for manipulating the parameter values are available by right clicking anywhere on a model parameter editor tab. If you right-click and select **Add to notes**, BioWin will place a tab-delimited text version of the currently selected model parameter tab into the **Simulation Notes** editor. This is useful for keeping records of model parameter values used in different simulation runs.

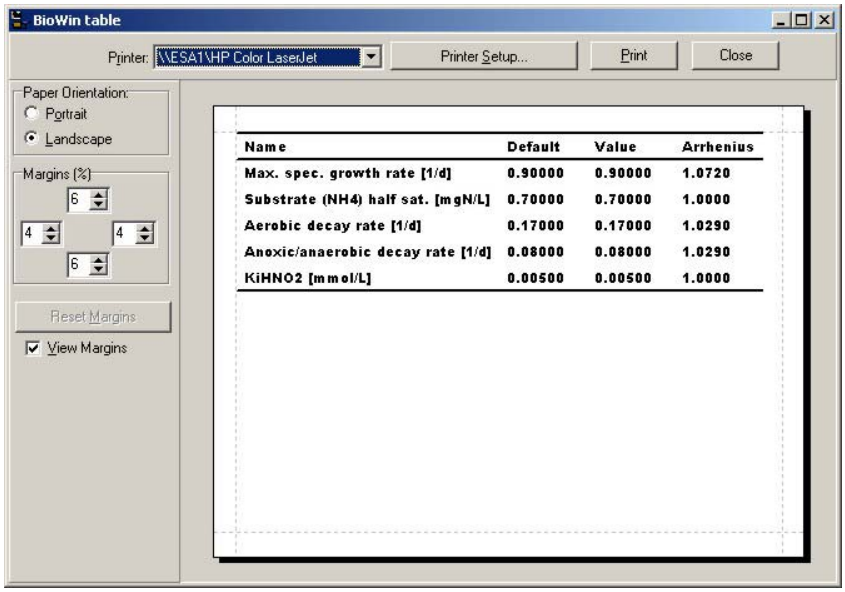
If you right-click and select **Copy** from the resulting popup menu, the contents of the tab (including column headings, the values in non-editable cells, and the values in editable cells) will be copied to the Windows clipboard. The clipboard contents then may be pasted into another application such as a word processor (all values will be separated by tabs) or a spreadsheet (each value will be placed in a separate cell).

You also may paste data from the clipboard into a model parameter editor by right-clicking and selecting **Paste** from the resulting popup menu.

Note : The cell that currently is selected in the model parameter editor corresponds to the first value in the first column of the data that you are pasting from the clipboard.

If BioWin does not recognize the format of the data that you are attempting to paste in, or if you select **Paste special** or **Paste from file**, the import wizard will be opened.

If you want to print the model parameter editor values in tabular form, then you can right-click and select **Print** from the resulting popup menu. When you do so, you will be presented with the following dialog box.



Dialog box used for printing model parameters

Use the **Printer** drop list box to select the printer you want to use for printing. The **Printer Setup…** button will open the printer setup dialog box which will allow you to access the printer's properties, set paper size, page orientation, and a number of other printer options (the options presented to you will be dependant on the printer you have). The **Print** button will send the print job to the printer and the printout will match the preview, which is shown. The **Close** button closes this dialog box and returns you to the Model Parameter Editor.

Using the **Paper Orientation** group, specify whether you want the printing to be done on a **Portrait** or **Landscape** page. The print preview gives you an idea of what the printout will look like under each format.

If you do not wish to see the size of the margins for your print job, you may de-select the box labeled **View Margins**. You can control the margins using three different methods:

Resize Cursors

Drag Print Object

54. Using the **Margins (%)** spin edits, you can adjust each margin as you like. The four spin edit boxes each control the margin that shares its position, that is, the top spin edit controls the top margin, the bottom spin edit controls the bottom margin, and so on. When you change a value, you will see changes in the print preview accordingly.
55. You may drag each margin using the mouse. Position the mouse cursor over the margin you wish to adjust until the horizontal or vertical resize cursor appears. Click the mouse button, hold it, and drag the margin to the position you wish it to occupy. Notice that when you finish dragging it, the values in the **Margins (%)** Spin Edits will have been updated.
56. By moving the object to be printed around on the page. When the mouse cursor takes the form of a hand, you may click and drag the entire object around on the page until it is in the desired position. Notice that when you finish dragging it, the values in the **Margins (%)** spin edits will have been updated

If after applying any one of these methods of adjusting margins you wish to reset the margins to the default values, you may do so by clicking the **Reset Margins** button.

Note: If you are having trouble with a print preview not fitting into the margins, ensure that you have a True Type font (font styles with a TT after their name) selected in your **Project|Current Project Options** - **Drawing board options** tab. If you want to print all of the tabs in the model parameter editor, click the **Print all** button at the bottom of the model parameter editor dialog box as opposed to right-clicking and selecting **Print** from the resulting popup menu on each individual model parameter editor tab.

Itinerary Editors

A number of elements have properties that may be entered as scheduled patterns rather than constant values. In this section, the use of the interfaces (called itinerary editors in BioWin) for entering these schedules is discussed.

A number of options for manipulating the itinerary data are available by right clicking anywhere on the itinerary tab. If you right-click and select **Add to notes**, BioWin will place a tab-delimited text version of the currently selected model parameter tab into the **Simulation Notes** editor. This is useful for keeping track of changes you make when doing multiple simulation runs on a configuration.

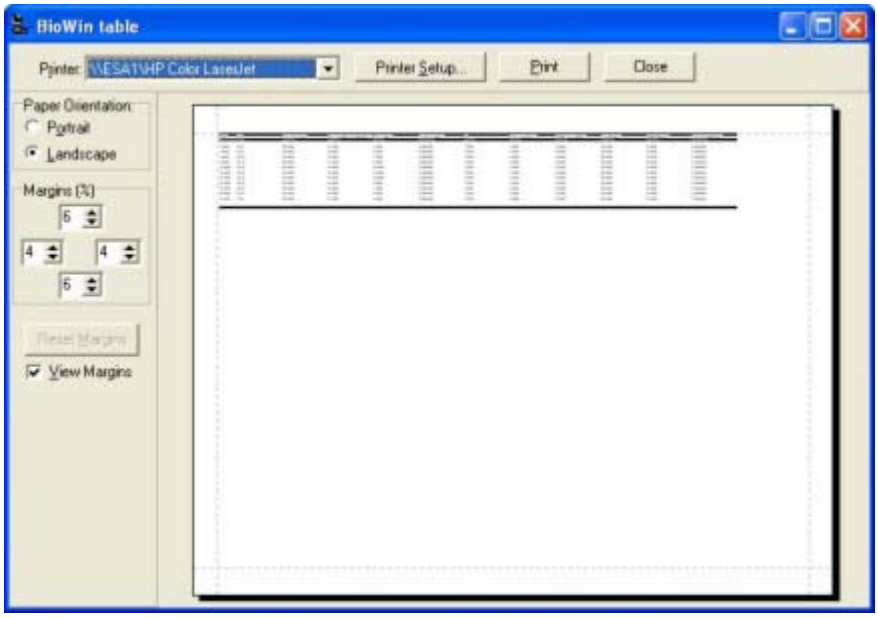
If you right-click and select **Copy** from the resulting popup menu, the contents of the tab (including column headings, the values in non-editable cells, and the values in editable cells) will be copied to the clipboard. The clipboard contents then may be pasted into another application such as a word processor (all values will be separated by tabs) or a spreadsheet (each value will be placed in a separate cell).

You also may paste data from the clipboard into an itinerary by right-clicking and selecting **Paste** from the resulting popup menu.

Note : The cell that currently is selected in the itinerary editor corresponds to the first value in the first column of the data that you are pasting from the clipboard.

If BioWin does not recognize the format of the data that you are attempting to paste in, or if you select **Paste special** or **Paste from file**, the import wizard will be opened.

If you want to print the itinerary in tabular form, then you can right-click on the itinerary tab and select **Print** from the resulting popup menu. When you do so, you will be presented with the following dialog box.



The print itinerary dialog

Use the **Printer** drop list box to select the printer you want to use for printing. The **Printer Setup…** button will open the printer setup dialog box which will allow you to access the printer's properties, set paper size, page orientation, and a number of other printer options (the options presented to you will be dependant on the printer you have). The **Print** button will send the print job to the printer and the printout will match the preview, which is shown. The **Close** button closes this dialog box and returns you to the Itinerary Editor.

Using the **Paper Orientation** group, specify whether you want the printing to be done on a **Portrait** or **Landscape** page. The print preview gives you an idea of what the printout will look like under each format.

If you do not wish to see the size of the margins for your print job, you may de-select the box labeled **View Margins**. You can control the margins using three different methods:

Resize Cursors

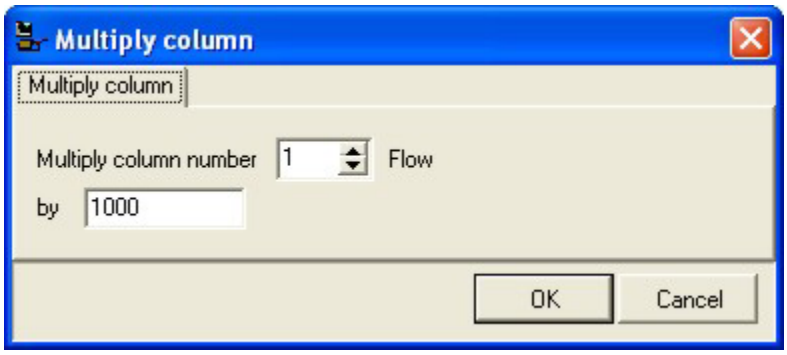
Drag Print Object

57. Using the **Margins (%)** spin edits, you can adjust each margin as you like. The four spin edit boxes each control the margin that shares its position, that is, the top spin edit controls the top margin, the bottom spin edit controls the bottom margin, and so on. When you change a value, you will see changes in the print preview accordingly.
58. You may drag each margin using the mouse. Position the mouse cursor over the margin you wish to adjust until the horizontal or vertical resize cursor appears. Click the mouse button, hold it, and drag the margin to the position you wish it to occupy. Notice that when you finish dragging it, the values in the **Margins (%)** Spin Edits will have been updated.
59. By moving the object to be printed around on the page. When the mouse cursor takes the form of a hand, you may click and drag the entire object around on the page until it is in the desired position. Notice that when you finish dragging it, the values in the **Margins (%)** spin edits will have been updated

If after applying any one of these methods of adjusting margins you wish to reset the margins to the default values, you may do so by clicking the **Reset Margins** button.

Note: If you are having trouble with a table print preview not fitting into the margins, ensure that you have a True Type font (font styles with a TT after their name) selected in your **Project|Options Drawing board options** tab.

If you wish to apply a multiplication factor to one of the columns in the itinerary, right-click on the itinerary and select **Multiply a column** from the resulting popup menu. When you do this, you will be presented with the following dialog box.

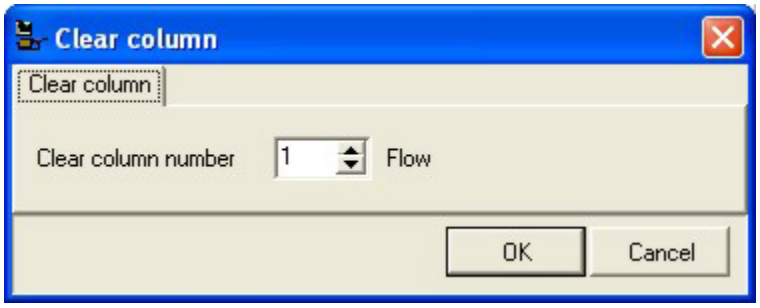


Utility for multiplying a column of an itinerary

Using the **Multiply column number** spin edit box, select the column that you wish to apply the multiplication factor to.

Note : As you change the value in this spin edit box, the label beside it changes to the heading of the selected column so you know exactly which values you will be changing. Once you have selected the column, then enter the multiplication factor in the **by** text edit box. To complete the multiplication operation, click **OK**.

If you wish to clear a column in an itinerary, right-click on the itinerary and select **Clear a column** from the resulting popup menu. When you do this, you will be presented with the following dialog box.



Utility for clearing a column of an itinerary

Using the **Clear column number** spin edit box, select the column that you wish to clear.

Note : As you change the value in this spin edit box, the label beside it changes to the heading of the selected column so you know exactly which values you will be changing. Once you have selected a column, click **OK** to clear it.

The procedures outlined above are common to all itinerary editors in BioWin. For the purpose of this manual, itinerary editors are divided into two broad classes:

1. **Standard Itineraries**
2. **Special Itineraries**

Details and examples of these two classes are given in the following sections.

Standard BioWin Itineraries

The following points highlight the use of standard itineraries. In the following sections, examples of standard itineraries are given.

- The timed pattern may span minutes, days, or even months, depending on the length of the simulation period.
- You can enter an influent pattern using the spreadsheet provided on this tab; start times may be entered in the first column, flow rates in the second column.
- To enter or change a value, click on the cell you would like to modify and enter a new value, or click on the number in the cell to edit that value. When you are satisfied with the value in a cell you may press the **Enter** (or **Return**) key on your keyboard, or click in another cell. You can use the arrow keys to move from one cell to the next; click the right mouse button to view a list of editing options.
- The **Cycle time** is specified using an edit box; this is the duration of the pattern, and must be specified in so that the simulator knows when to start repeating the cycle.

Note : If you have an event outside of the range of your cycle time, a warning will be displayed in red text at the bottom of the itinerary and you will not be able to close the itinerary until you rectify this. You also may specify a **Cycle offset**; this will have the effect of offsetting your timed pattern from the start time of the simulation - that is, you "step into" your timed pattern by an amount equal to the cycle offset.

- **Time units** may be selected using the radio buttons; you can specify days, hours, or minutes. This tab also contains a group of radio buttons for specifying **Flow units**; there are four different unit options: L/d, ML/d, m3/d, and mgd (US).
- You can increase or decrease the number of **Rows** (i.e. intervals) in a timed pattern using the spin edit box.
- If there are blanks in your time column, you can click on the check box to **Interpolate blank time cells** (all influent specifications require associated time values). There are a number of options for replacing blanks in the other columns. You may choose from one of the following **Blank fill styles**: the last value or zero, a time-weighted average value, an interpolated value, or an average value. Blank values are interpolated by default.
    1. If you select **Last value or zero**, then a blank cell will be filled with the value that is contained in the previous non-blank cell. If BioWin finds no values in a column, then the column will be filled with zeroes. This way, if you want a column filled with zeroes, you don't have to enter a zero in the first cell.

2. If you select Time weighted average value, BioWin will use the column's non-blank, non-zero cells to compute a time-weighted average. This calculated time-weighted average will then be placed in all blank cells.
3. If you select Interpolated value, BioWin will fill the blank cell (or groups of contiguous blank cells) with a linearly interpolated value. The linear interpolation is based only on the points immediately adjacent to the blank cell(s).

Note : If there are no values in the column and this option is selected, the column will be filled with zeroes.

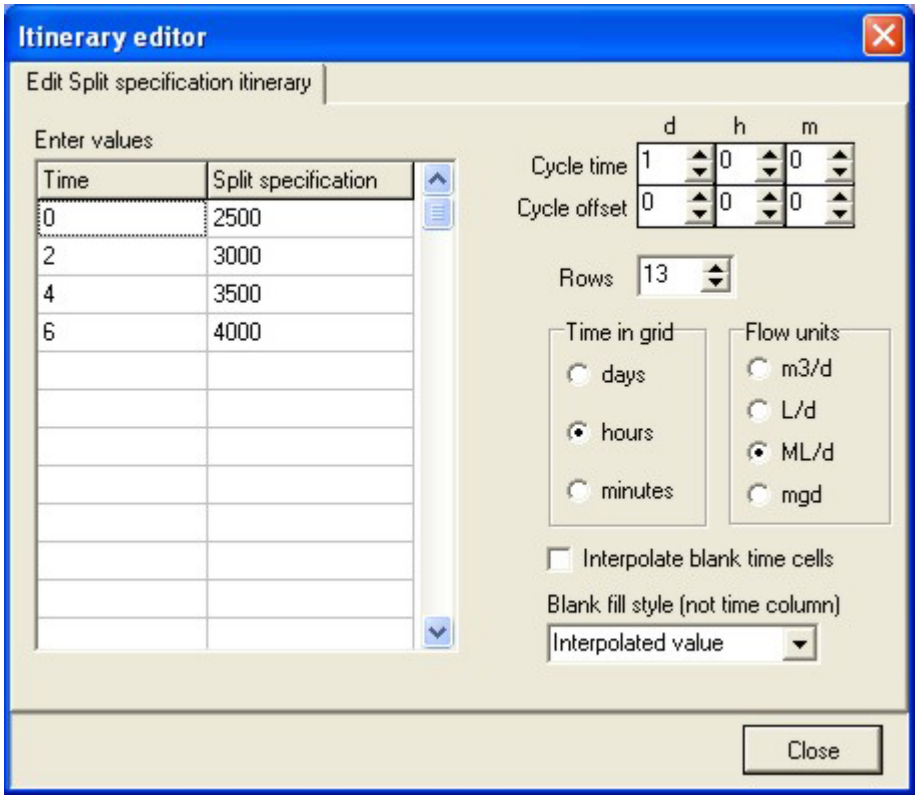
4. If you select Average value, BioWin will use the column's non-blank, non-zero cells to compute a straight arithmetic average. This calculated average will then be placed in all blank cells.

As outlined earlier, a number of options for copying and pasting data, printing the itinerary, and multiplying column values are available by right clicking on the itinerary tab.

Note: Use care when switching back and forth between constant and scheduled influents. BioWin assigns the constant values to the first row of your schedule, and vice versa. So in the case where you are switching between a constant influent that is *different* from the first row of your scheduled influent, ensure that the values are what you want them to be before running simulations.

Split Itinerary

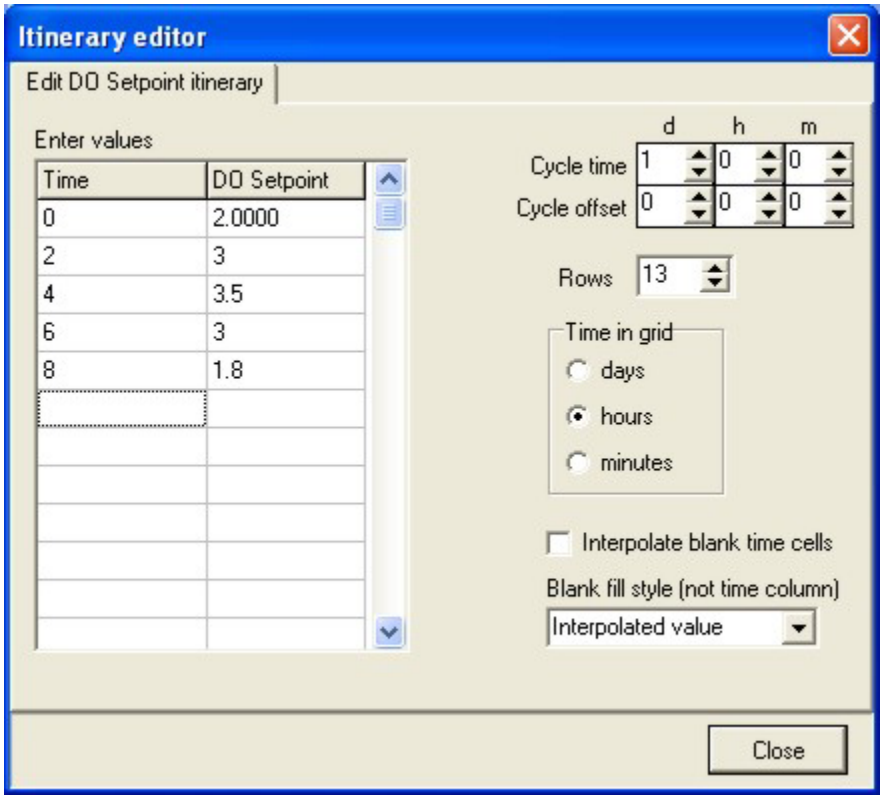
This itinerary editor, shown below, allows the user to enter a scheduled pattern for an element's split method. This dialog box applies to elements that split flows such as splitters, settlers (primary and secondary), grit tanks, and dewatering elements.



The split itinerary editor

DO Setpoint Itinerary

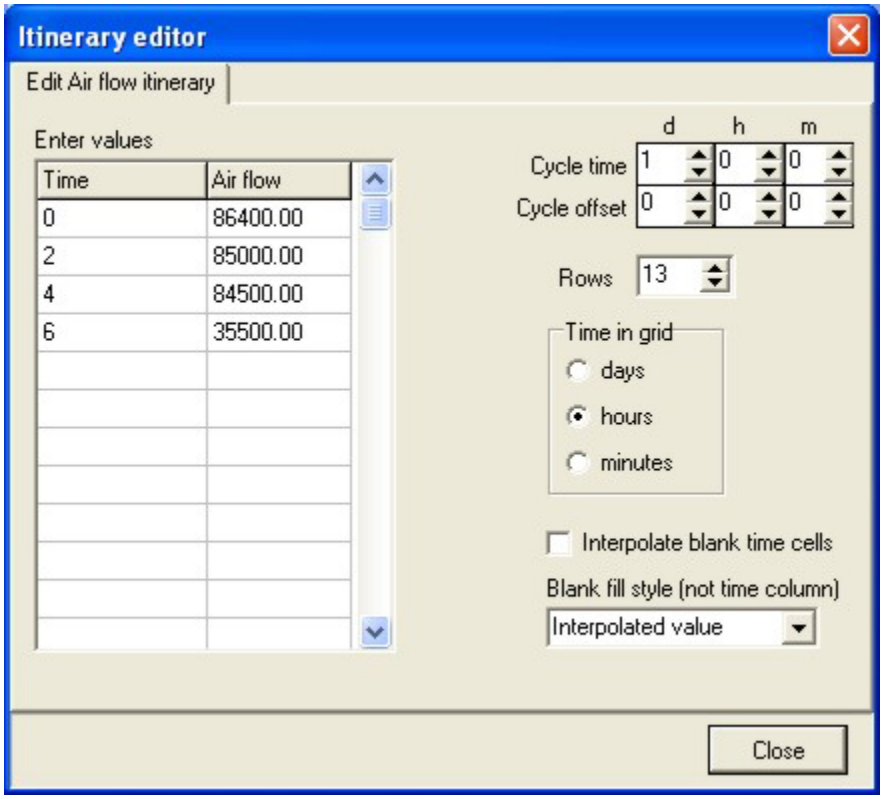
This itinerary editor, shown below, allows the user to enter a scheduled pattern for an element's DO setpoint. This dialog box applies to aerated elements such as bioreactors and aerobic digesters.



A DO setpoint itinerary editor

Air Flowrate Itinerary

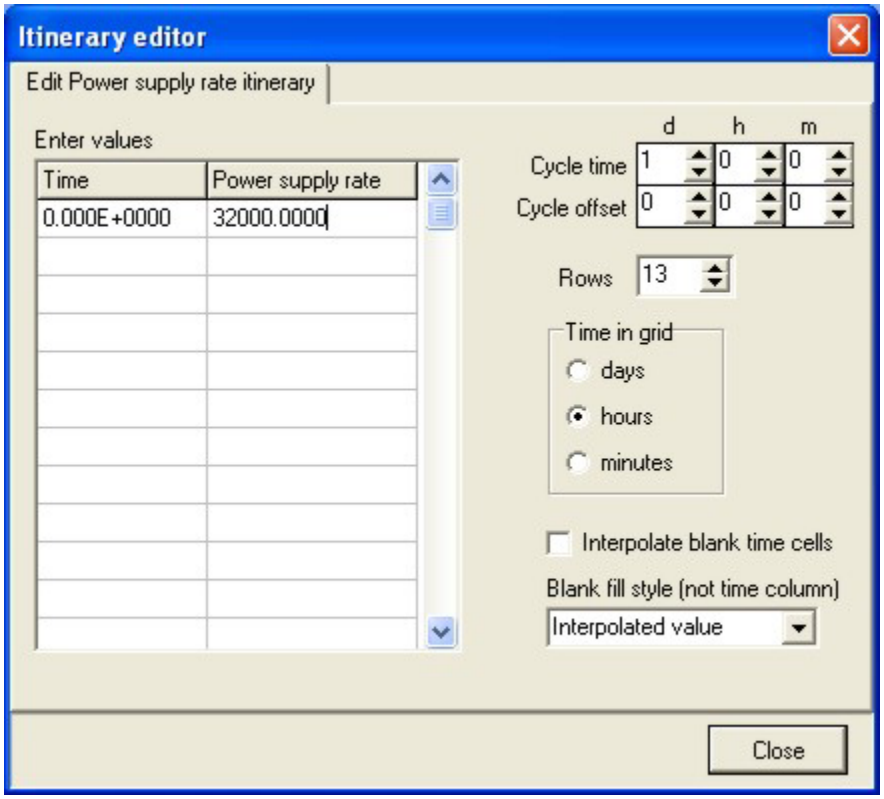
This itinerary editor, shown below, allows the user to enter a scheduled pattern for an element's air flowrate. This dialog box applies to aerated elements such as bioreactors and aerobic digesters.



An air flowrate itinerary editor

Power Supply Rate Itinerary

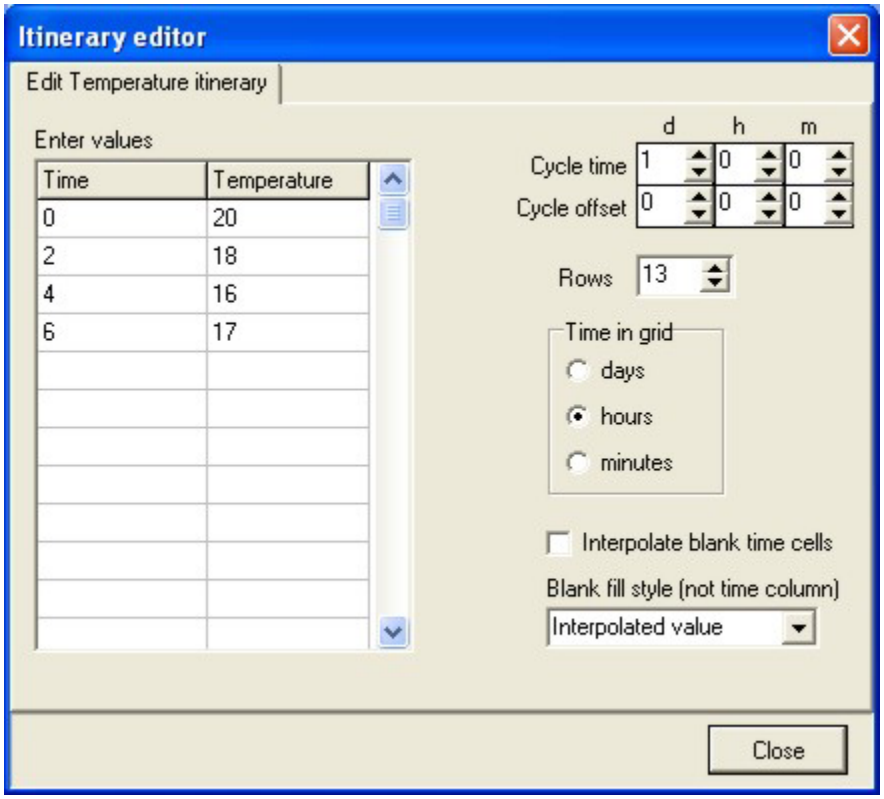
This itinerary editor, shown below, allows the user to enter a scheduled pattern for an element's power supply rate. This dialog box applies to surface aerated elements such as surface aerator bioreactors and brush aerator bioreactors.



An power supply rate itinerary editor

Temperature Itinerary

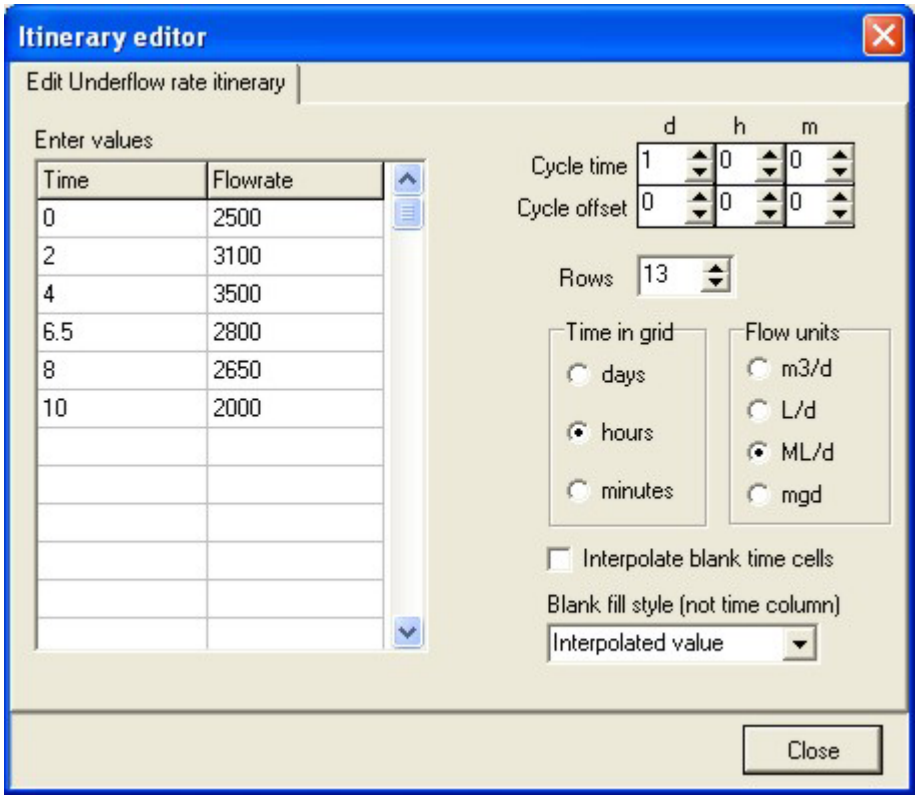
This itinerary editor, shown below, allows the user to enter a scheduled pattern for an element's temperature. This dialog box applies to elements incorporating temperature such as bioreactors, SBRs and aerobic digesters.



A temperature itinerary editor

Underflow Rate Itinerary

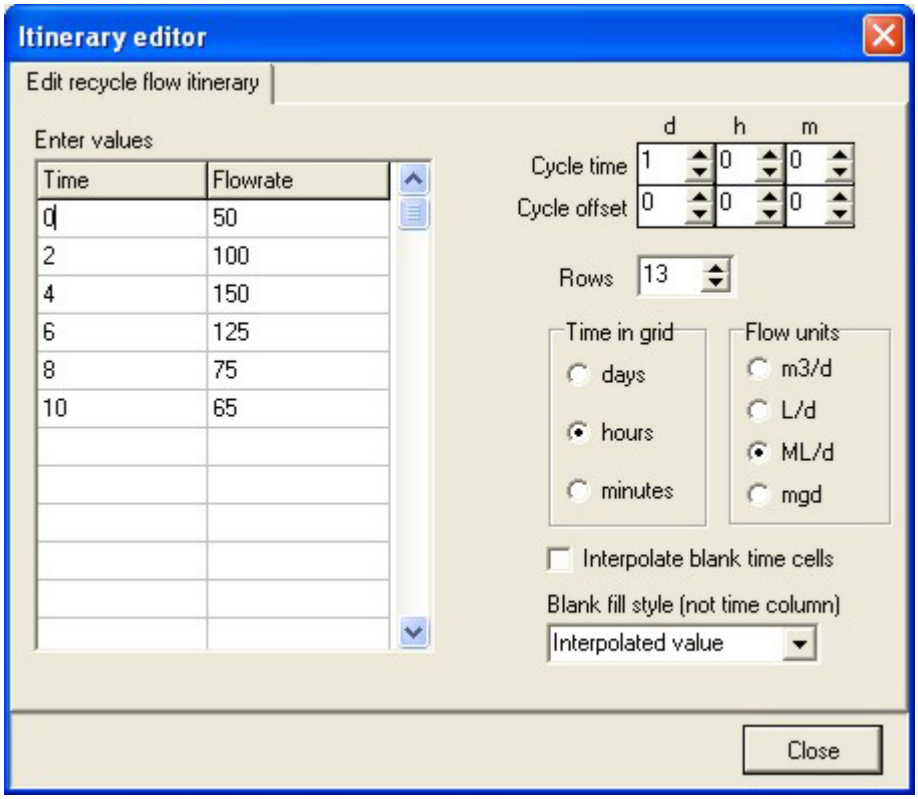
This itinerary editor, shown below, allows the user to enter a scheduled pattern for an element's (e.g. an SBR or clarifier) liquid underflow.



Dialog box for entering a liquid underflow pattern

Internal Recycle Flowrate Itinerary

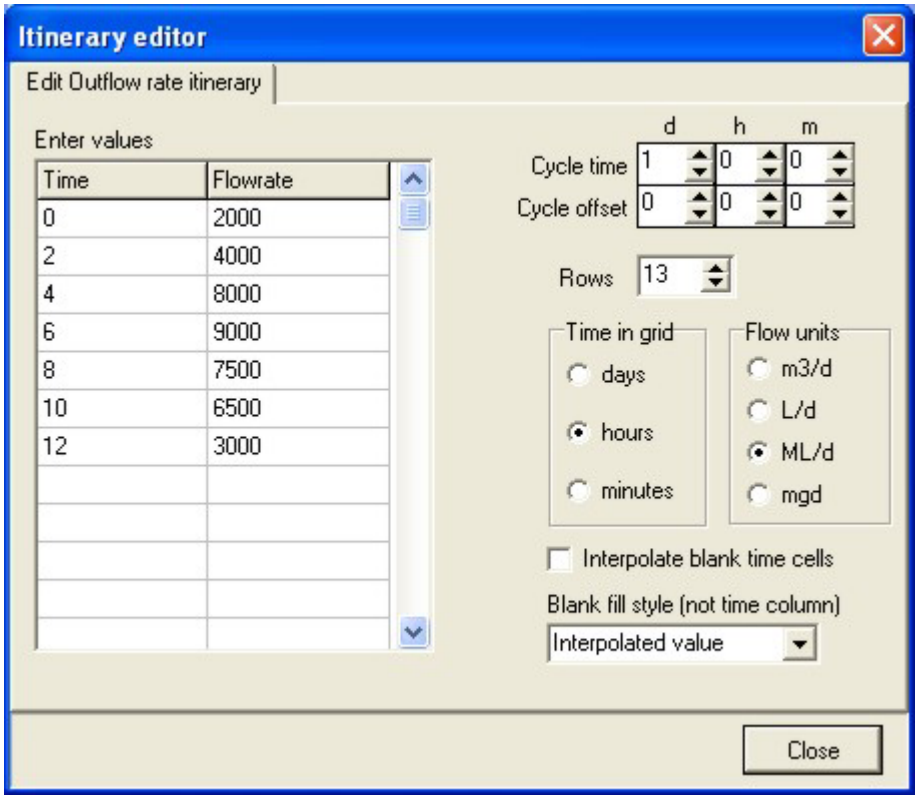
This itinerary editor, shown below, allows the user to enter a scheduled pattern for an element's (e.g. SBR + 2 always-mixed prezones) internal recycle flowrate.



Dialog box for entering an internal liquid recycle flowrate pattern

Liquid Outflow Itinerary

This itinerary editor, shown below, allows the user to enter a scheduled pattern for an element's (e.g. a variable volume/batch bioreactor) liquid outflow.



Dialog box for entering a liquid outflow pattern

Special BioWin Itineraries

The following points highlight the use of special itineraries. In the following sections, examples of special itineraries are given, along with the distinguishing feature that makes them different from standard itineraries.

- The timed pattern may span minutes, days, or even months, depending on the length of the simulation period.
- You can enter an influent pattern using the spreadsheet provided on this tab; start times may be entered in the first column, flow rates in the second column.
- To enter or change a value, click on the cell you would like to modify and enter a new value, or click on the number in the cell to edit that value. When you are satisfied with the value in a cell you may press the **Enter** (or **Return**) key on your keyboard, or click in another cell. You can use the arrow keys to move from one cell to the next; click the right mouse button to view a list of editing options.
- The **Cycle time** is specified using an edit box; this is the duration of the pattern, and must be specified in so that the simulator knows when to start repeating the cycle.

Note : If you have an event outside of the range of your cycle time, a warning will be displayed in red text at the bottom of the itinerary and you will not be able to close the itinerary until you rectify this. You also may specify a **Cycle offset**; this will have the effect of offsetting your timed pattern from the start time of the

simulation - that is, you "step into" your timed pattern by an amount equal to the cycle offset.

- **Time units** may be selected using the radio buttons; you can specify days, hours, or minutes. This tab also contains a group of radio buttons for specifying **Flow units**; there are four different unit options: L/d, ML/d, m3/d, and mgd (US).
- You can increase or decrease the number of **Rows** (i.e. intervals) in a timed pattern using the spin edit box.
- If there are blanks in your time column, you can click on the check box to **Interpolate blank time cells** (all influent specifications require associated time values). There are a number of options for replacing blanks in the other columns. You may choose from one of the following **Blank fill styles**: the last value or zero, a time-weighted average value, an interpolated value, or an average value. Blank values are interpolated by default.

    1. If you select **Last value or zero**, then a blank cell will be filled with the value that is contained in the previous non-blank cell. If BioWin finds no values in a column, then the column will be filled with zeroes. This way, if you want a column filled with zeroes, you don't have to enter a zero in the first cell.
    2. If you select Time weighted average value, BioWin will use the column's non-blank, non-zero cells to compute a time-weighted average. This calculated time-weighted average will then be placed in all blank cells.
    3. If you select Interpolated value, BioWin will fill the blank cell (or groups of contiguous blank cells) with a linearly interpolated value. The linear interpolation is based only on the points immediately adjacent to the blank cell(s).

Note : If there are no values in the column and this option is selected, the column will be filled with zeroes.

    4. If you select Average value, BioWin will use the column's non-blank, non-zero cells to compute a straight arithmetic average. This calculated average will then be placed in all blank cells.

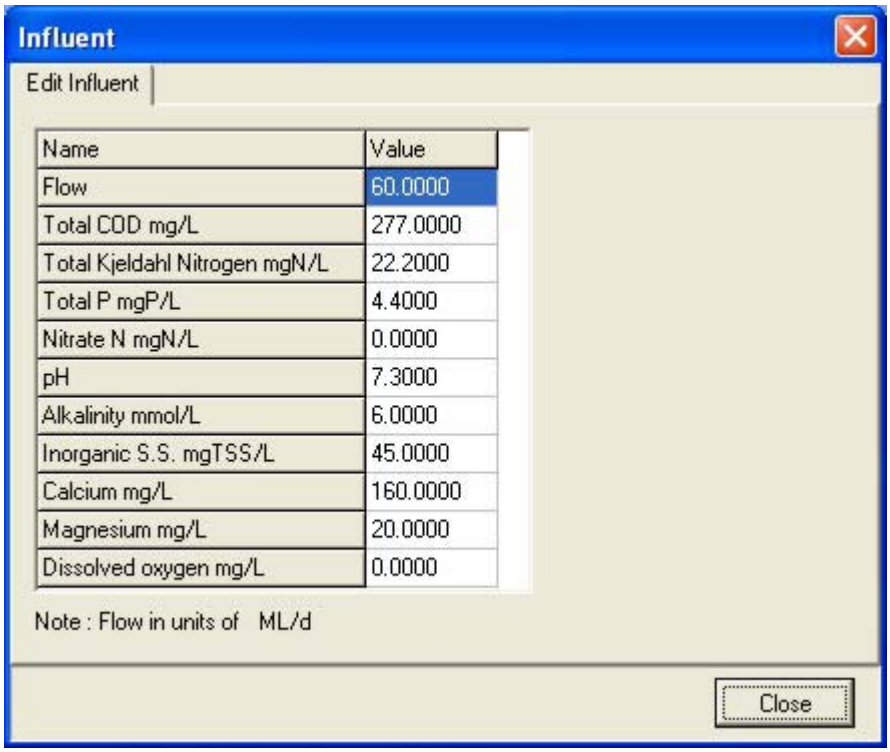
As outlined earlier, a number of options for copying and pasting data, printing the itinerary, and multiplying column values are available by right clicking on the itinerary tab.

Note: Use care when switching back and forth between constant and scheduled influents. BioWin assigns the constant values to the first row of your schedule, and vice versa. So in the case where you are switching between a constant influent that is *different* from the first row of your scheduled influent, ensure that the values are what you want them to be before running simulations.

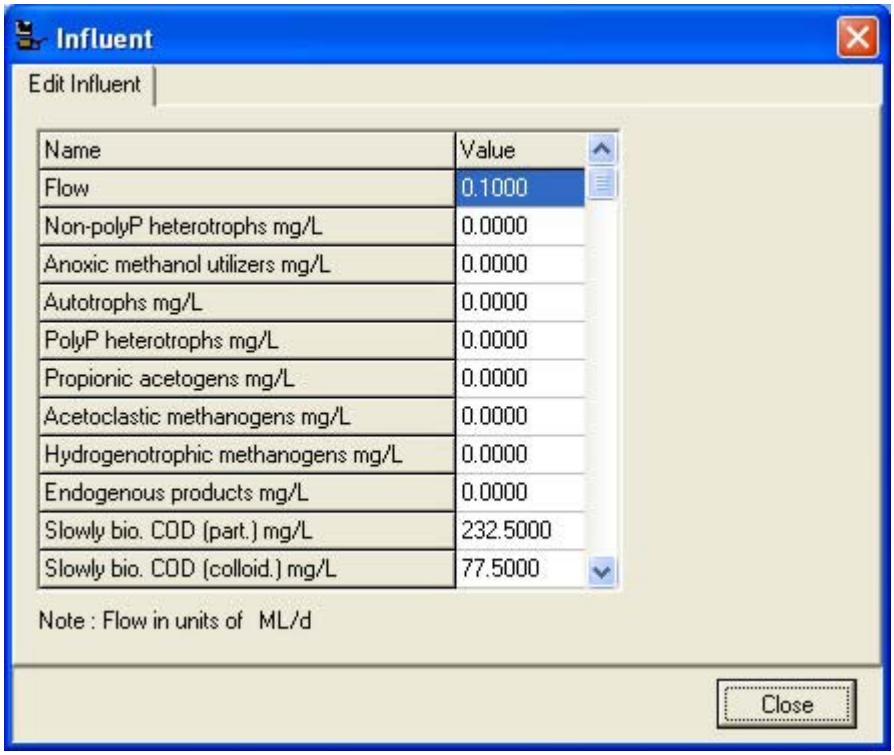
Influent Itinerary

The appearance of the influent itinerary editor depends on whether the influent type is constant or variable, and whether the influent element specifies wastewater composition in terms of totals and fractions (i.e. a standard influent), state variable concentrations (i.e. a state variable influent), a BOD influent, a methanol input, or a

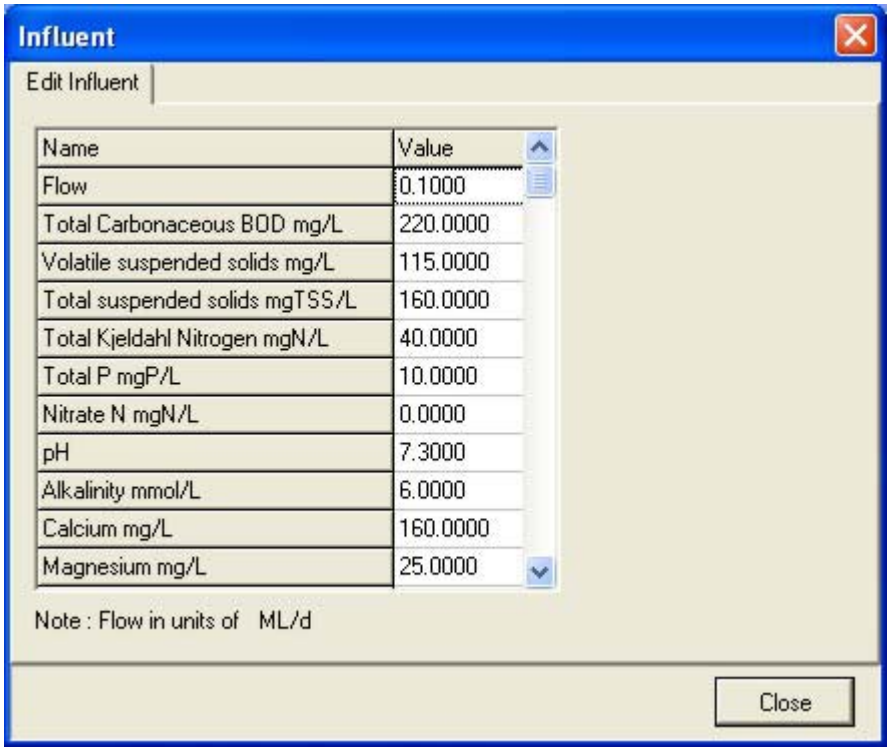
metal addition influent. If the influent is constant, then the influent data are entered in the simple dialogs shown below.



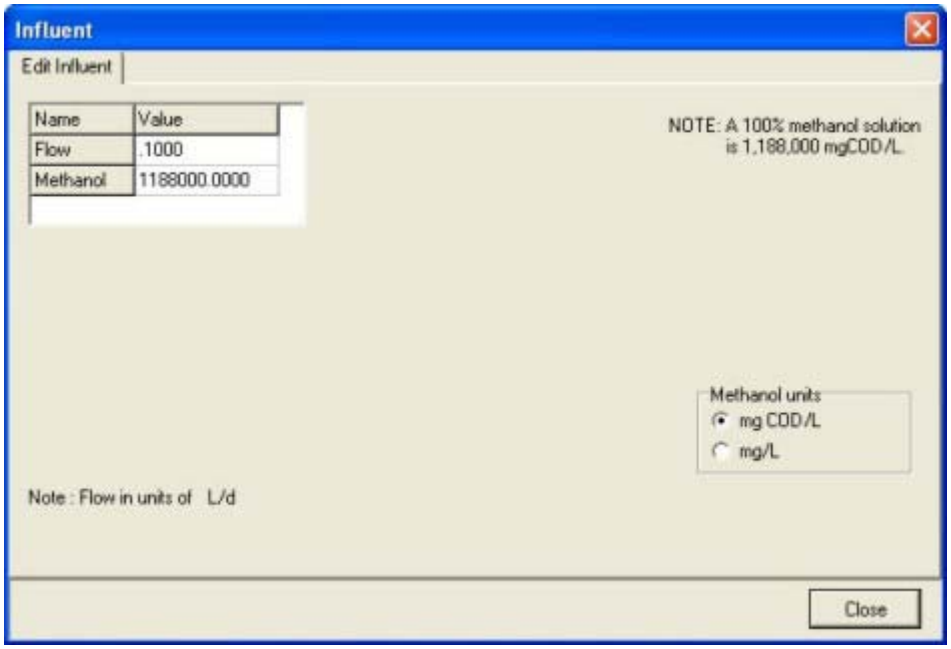
A constant influent itinerary



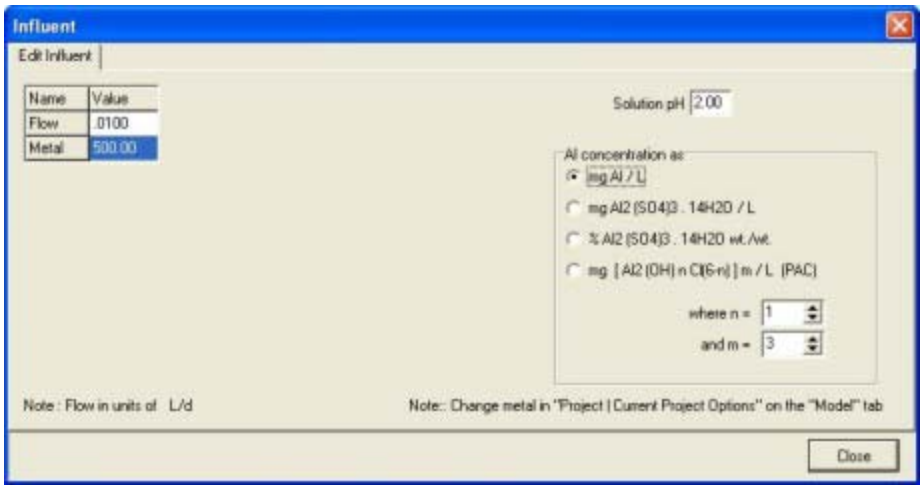
A constant state variable influent itinerary



A constant BOD influent itinerary

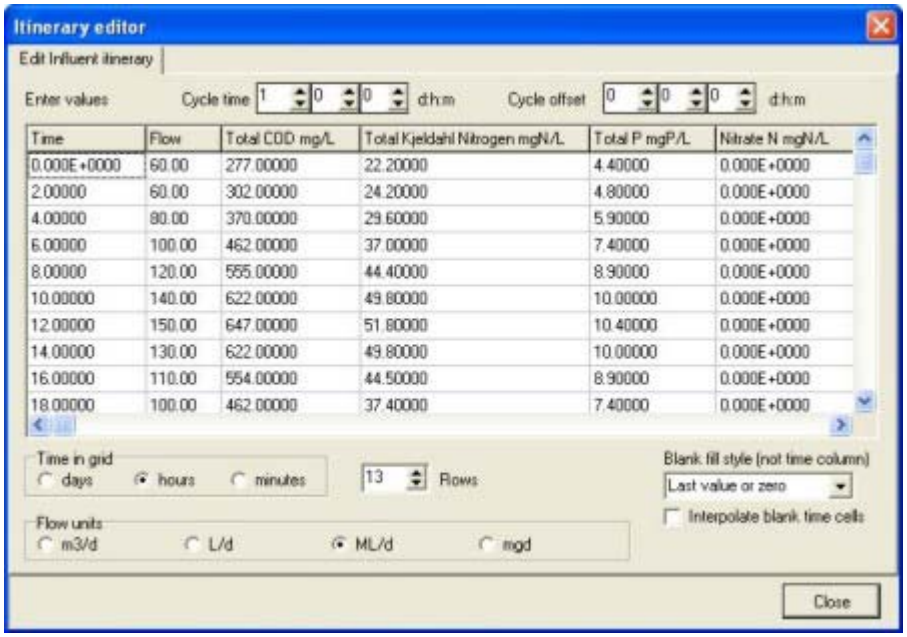


A constant methanol influent itinerary

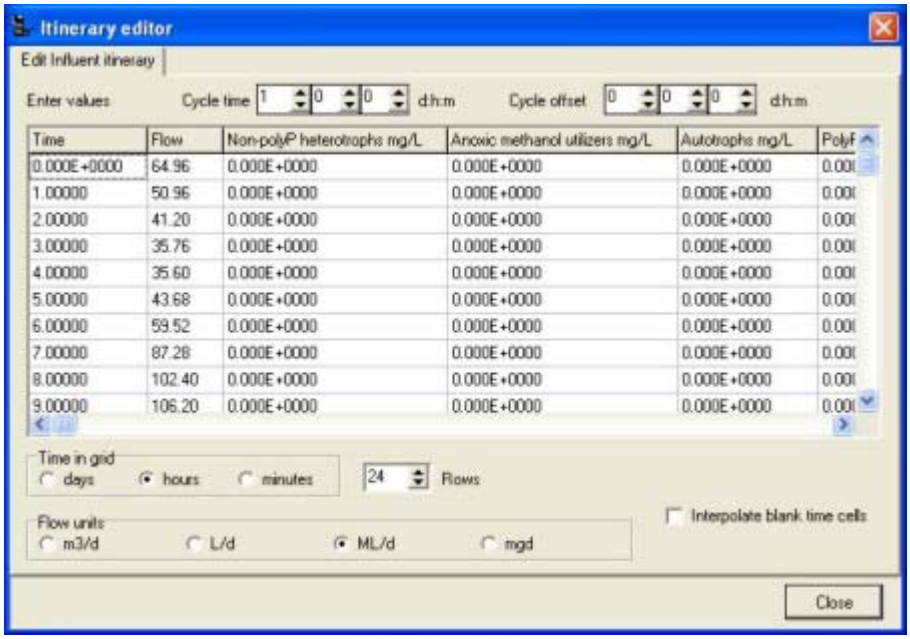


A constant metal addition influent itinerary

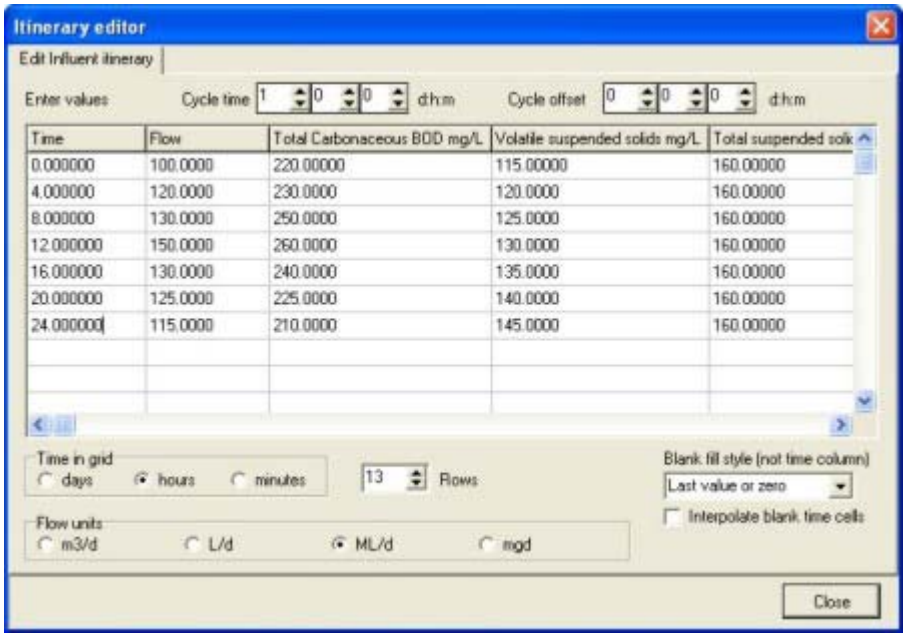
If the influent type is variable, then the influent itinerary will appear as shown below to allow the user to enter a scheduled pattern.



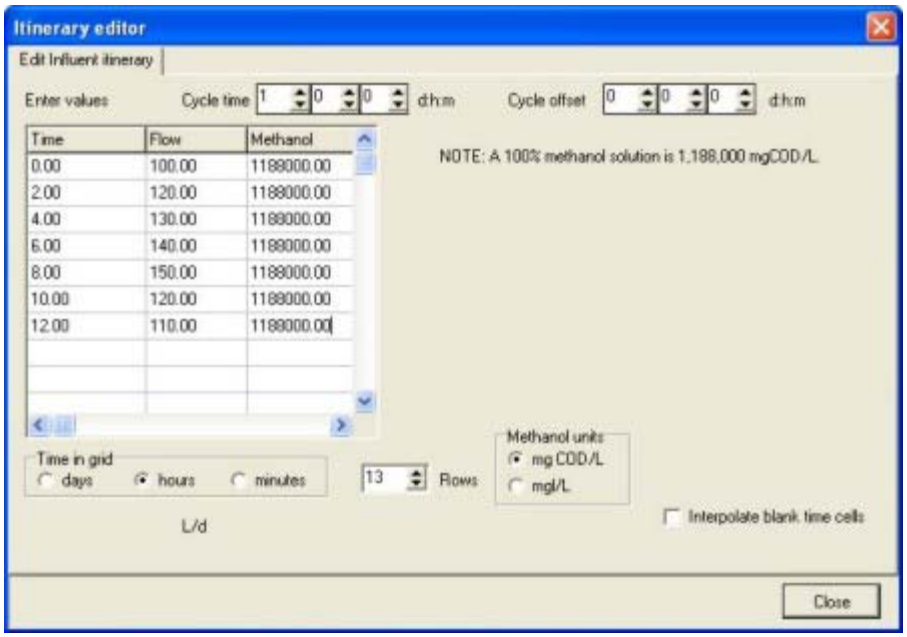
A variable influent itinerary



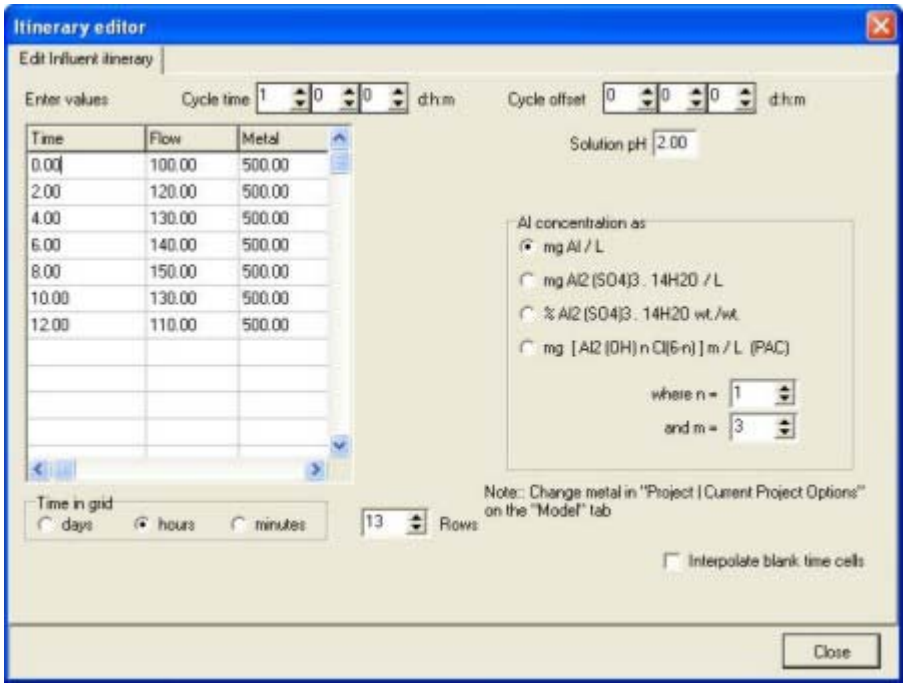
A variable state variable influent itinerary



A variable BOD influent itinerary



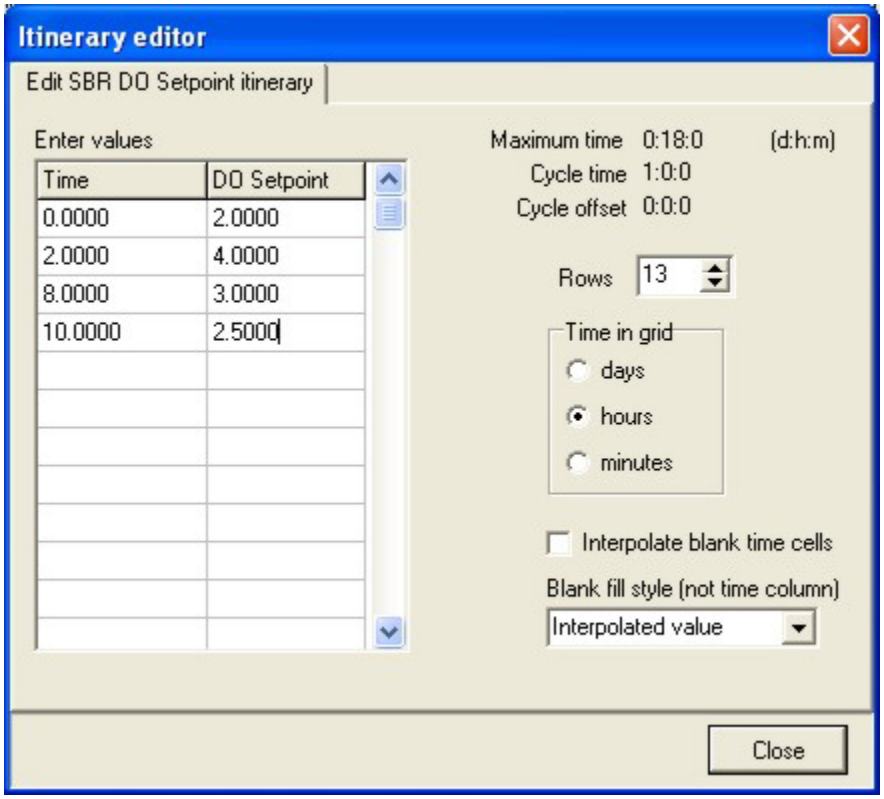
A variable methanol influent itinerary



A variable metal addition influent itinerary

SBR DO Setpoint Itinerary

This itinerary editor, shown below, allows the user to enter a scheduled pattern for an SBR element's DO setpoint.



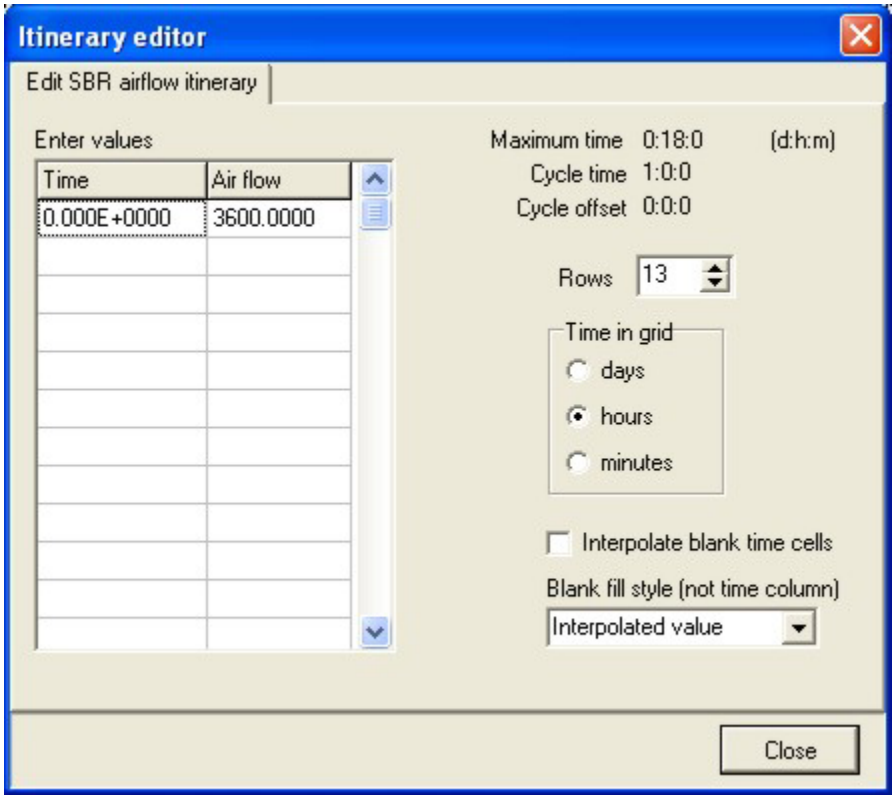
An SBR DO setpoint itinerary editor

Note : For the SBR DO setpoint itinerary, the **Cycle time** cannot be specified since it is linked to the SBR Operation cycle time. The itinerary editor provides you with feedback by telling you the **Maximum** time span in which you can aerate (this is set by the SBR Mixing/Aeration cycle length – you cannot have aeration occurring during the Settling/Decant phase).

Note : If you have an aeration event outside of the range of your maximum allowable time, a warning will be displayed in red text at the bottom of the itinerary and you will not be able to close the itinerary until you rectify this. Notice also that you may not have a different **Cycle offset** than that of your SBR operation cycle offset.

SBR Air Flowrate Itinerary

This itinerary editor, shown below, allows the user to enter a scheduled pattern for an SBR element's air flowrate.



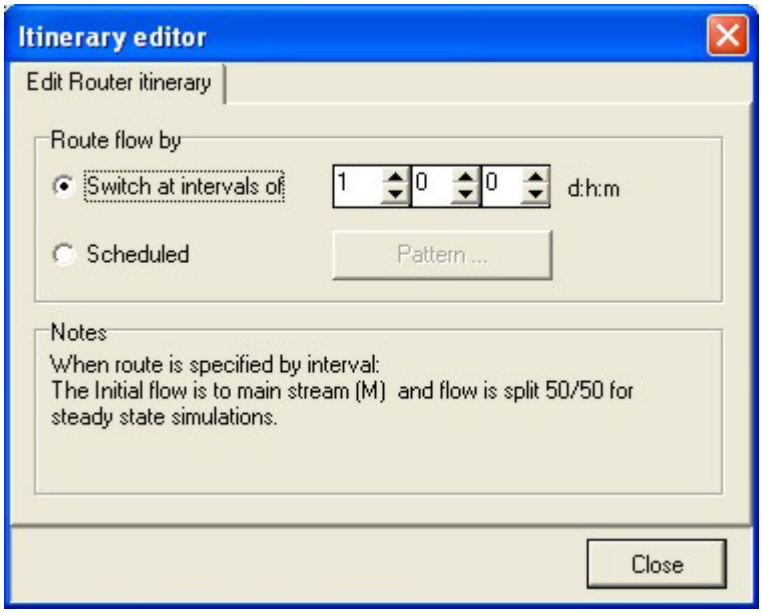
An SBR air flowrate itinerary editor

Note : For the SBR air flowrate itinerary, the **Cycle time** cannot be specified since it is linked to the SBR Operation cycle time. The itinerary editor provides you with feedback by telling you the **Maximum** time span in which you can aerate (this is set by the SBR Mixing/Aeration cycle length – you cannot have aeration occurring during the Settling/Decant phase).

Note : If you have an aeration event outside of the range of your maximum allowable time, a warning will be displayed in red text at the bottom of the itinerary and you will not be able to close the itinerary until you rectify this. Notice also that you may not have a different **Cycle offset** than that of your SBR operation cycle offset.

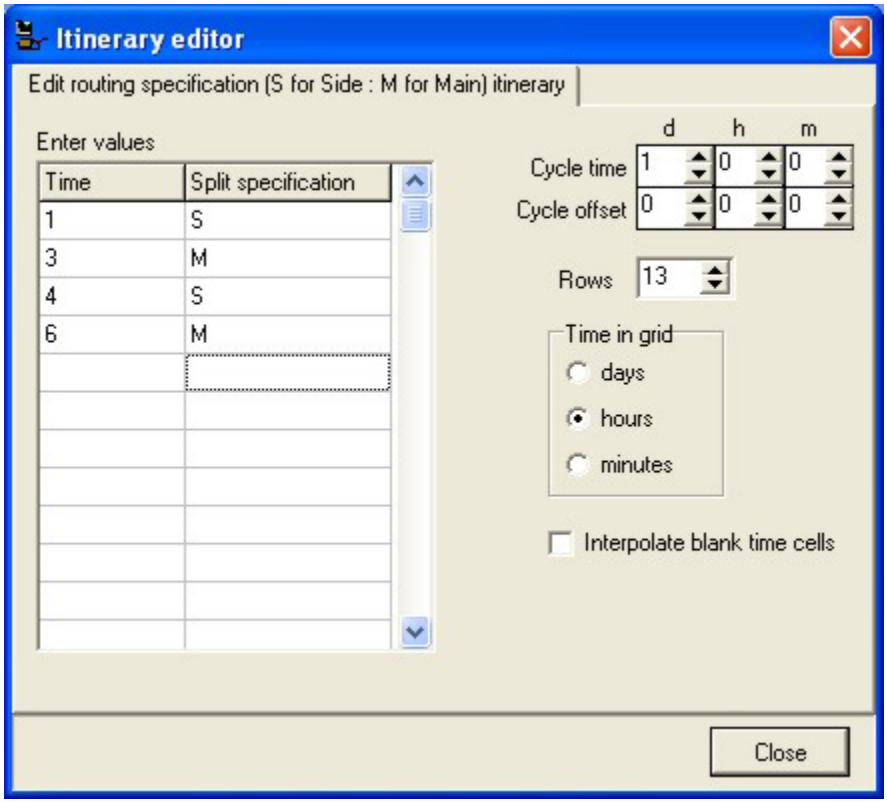
Router Itinerary

This itinerary editor, shown below, allows the user to specify a flow routing pattern for a splitter element.



A router itinerary for alternating flow routing

You may specify either to **Switch at intervals**, or you may use a **Scheduled pattern**. Click on the radio button to select the option you want. If you select to switch the flow at regular intervals, you may enter the interval length in the edit boxes (hours and minutes). If you choose a scheduled pattern, click on the **Pattern** button to open the router itinerary editor, shown below. The operation of this itinerary editor is the same as other standard itinerary editors.

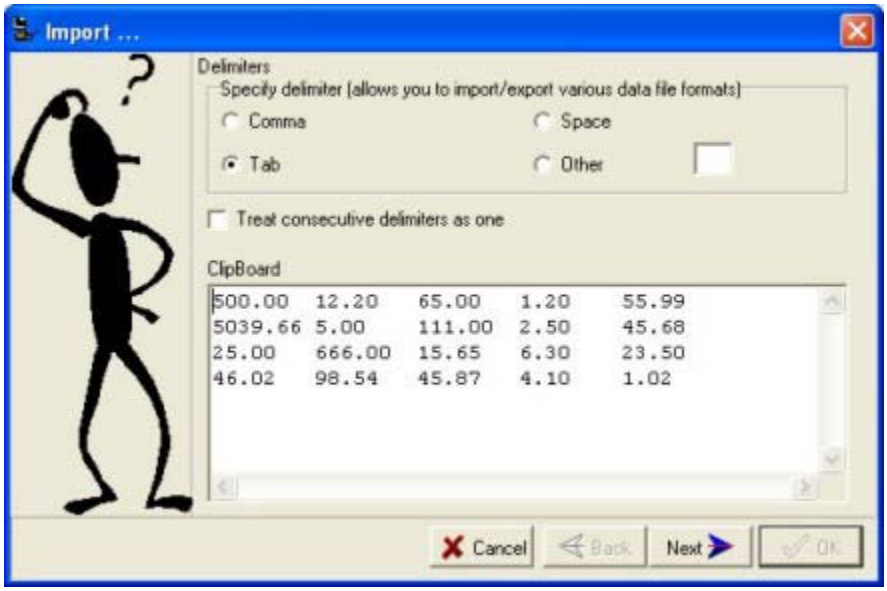


Dialog box for entering a router pattern

Import Wizard

In this section, the BioWin import wizard is discussed in detail. The import wizard is invoked when you paste data into BioWin from the Windows Clipboard or from a file. The import wizard takes you through the following steps:

1. In the first step, you specify the type of delimiters that separate individual data values in the group that you are pasting. From the **Specify delimiter** radio button group, select from **Comma**, **Space**, **Tab**, or **Other** (where *you* enter the delimiter that is used). The viewing area at the bottom of the dialog box that shows the contents of the file that you are pasting is meant to assist you in choosing the correct delimiter by displaying the delimiters in the file. The viewing area has a file size limit of 64 KB, so you will not be able to properly preview your file if it exceeds this limit. This does not compromise BioWin's ability to import your data if the file size is greater than 64 KB.

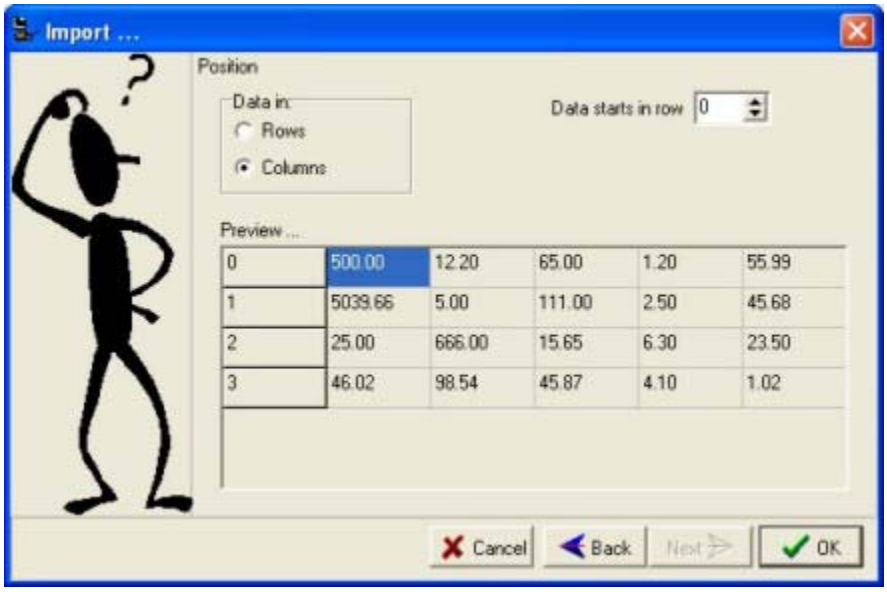


The first step of the import wizard

Note : If you have instances in your data file where you have data values separated by multiple delimiters, then you should check the box labeled **Treat consecutive delimiters as one**. This will avoid pasting gaps into the itinerary. This option will be enabled only when you have spaces or other delimiters in your data, to decrease the chances of this option causing BioWin to treat a missing data value as an extra delimiter, which would result in the deletion of the column containing the missing data value.

2. In the second step, you are shown a preview of what the data will look like when it is pasted into the itinerary. This is helpful to ensure that you have selected the proper options in the first step. If you are not satisfied with the way the data looks, you can return to the previous step by clicking the **Back** button. You can transpose the data you are pasting by switching between the **Rows** and **Columns** button in the

Data in: radio button group. The preview will change to reflect these changes. Also, if you wish to exclude the first row or several rows, you can increase the value in the **Data starts in row** spin edit box. Once again, the preview will change to reflect these changes.

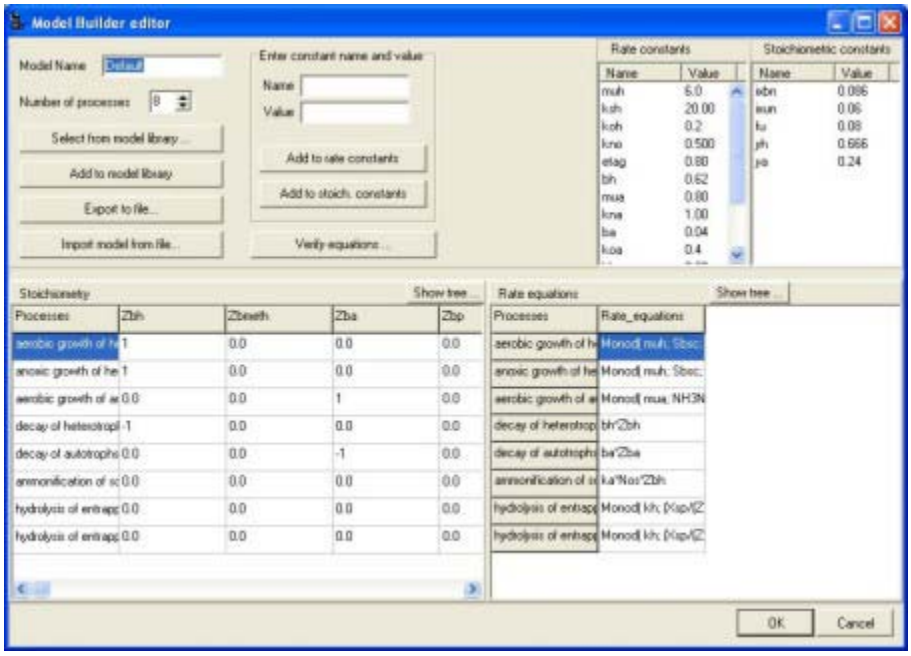


The second step of the import wizard

When you are happy with the preview of the data that will be pasted into the itinerary, click the **OK** button to finish the **Paste special** operation.

Model Builder

The model builder, shown in the picture below, is used to enter stoichiometry and rate equations for user-defined models. The model builder may be invoked by clicking the **Specify local model…** button (if the **Local Builder model** option is selected) on the **Operation** tab of the model builder element. Alternatively, the model builder may be invoked by selecting the **Model Builder…** command from the **Project** menu.



The model builder with activated sludge model number one (ASM 1) loaded

The model builder allows you to perform a number of tasks related to creating your own models:

- Enter stoichiometry and rate constants;
- Enter processes which act on the BioWin state variables;
- Enter stoichiometry and rate equations using a powerful equation editor;
- Manage your model(s) by allowing you to import and export to various file formats.

These tasks are explained in more detail in the following sections.

Entering Model Constants

You must assign your model constants a name and a value. To enter a model constant, type the name of the constant in the **Name** text edit area. Next, type the value you wish to assign to the constant in the **Value** text edit area.

When you are satisfied with the name and value you have assigned to your constant, you may enter it into your model as a rate constant by clicking the **Add to rate constants** button. When you do this, the constant will appear in the **Rate constants** list. To enter the constant into your model as a stoichiometry constant, click the **Add to stoich. constants** button. When you do this, the constant will appear in the **Stoichiometric constants** list.

Note : When you add a constant to one of the two lists, the constant name and value remain in their respective text edit areas. This makes it easy for you to enter the constant into both lists with the same name and value if you wish.

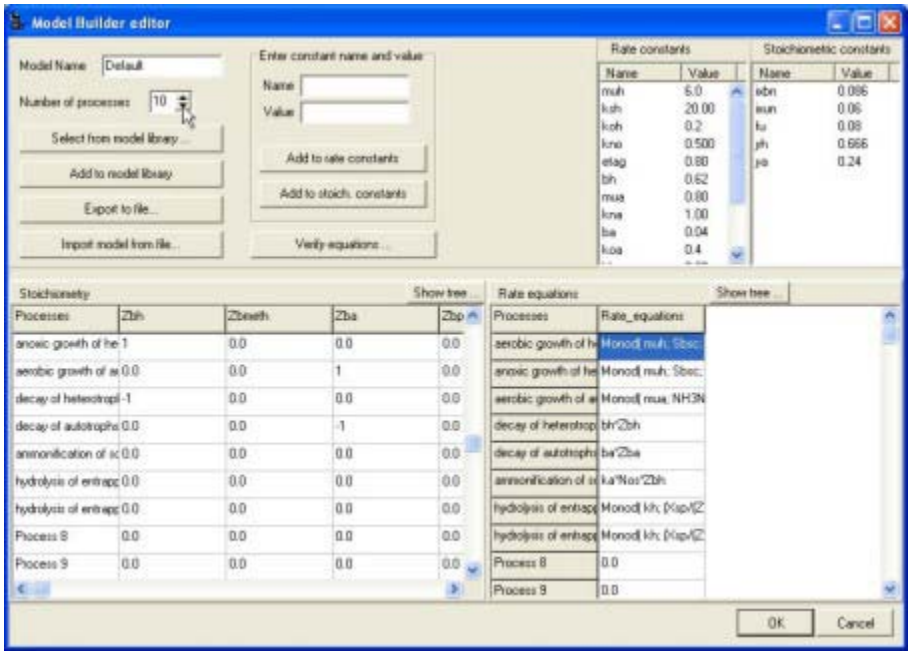
If you wish to change a model constant, click on it in the **Rate constants** list or in the **Stoichiometric constants** list so that it is highlighted blue. You can now

delete the constant if you wish by pressing the **Delete** key on your keyboard. You can also edit the name and/or value of the constant by editing the entry in the **Name** and/or **Value** text edit area, and pressing the appropriate **Add to…** button.

Entering Model Processes

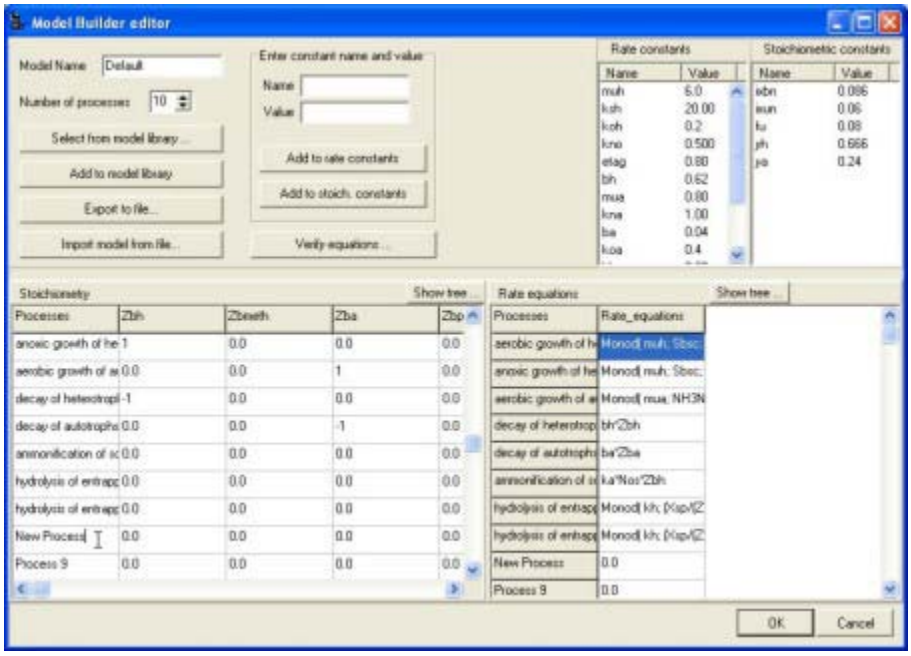
To add processes to your model, increase the value of the **Number of processes** spin edit box. When you do this, additional unnamed processes will be added to your model. In the example shown below, the unnamed processes **Process 3** and **Process 4** have been added to the model in the **Stoichiometry** and **Rate equations** sections.

Note : Values of 0 are entered for stoichiometry and rate equations.



Adding additional processes to your model

To assign the processes names, simply click on the process name in the **Stoichiometry** section of the editor so that it is highlighted blue, and type in the new name. Your changes will automatically be reflected in the **Rate equations** section. In the example below, the new process named **Process 3** has been renamed to a process called **Auto Conversion**.



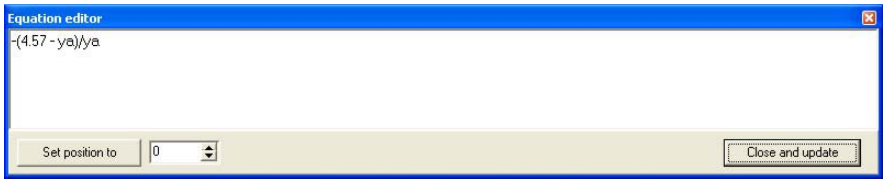
Changing the name of a process

Entering Model Equations

There are two methods for entering model equations. If the equation is small and straightforward, you may be able to enter it directly into the appropriate model cell. To do this, click in the stoichiometry or rate equation cell where you wish to enter the equation. When you do this, the text currently in the cell will be highlighted blue. Clicking in the blue highlighted area will present you with a cursor that will allow you to enter your equation. For more complex lengthy equations, you will probably find it easier to use the built in equation editor.

Invoking the Equation Editor

To invoke the equation editor, double-click in the stoichiometry or rate cell where you wish to enter an equation. This will invoke the **Equation editor**, shown in the picture below.



Equation editor showing the stoichiometric equation for the Dissolved Oxygen state variable for the autotroph growth process.

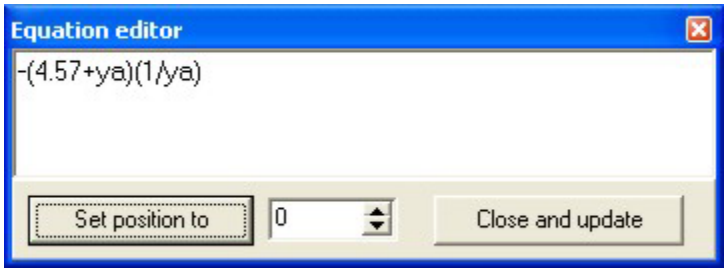
Before proceeding further with explanation of the equation editor, the following points on exiting should be noted:

1. To exit the equation editor **without accepting the changes you have made**, click the small **x** in the upper right hand corner of the equation editor window.
2. To exit the equation editor **and accept the changes you have made**, click the **Close and update** button.

You can use the **Set position to** button to quickly move the cursor around within the equation editor. The position refers to the number of spaces from the left (or beginning) of a line of text. In the example picture above, the cursor has been jumped to the sixth position.

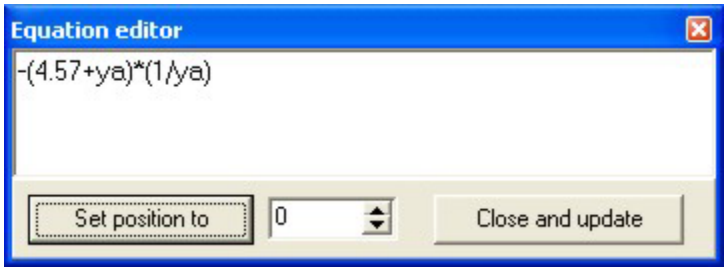
Equation Editor Syntax

The equation editor provides you with a large window into which you may enter your equation text. You must take care to use proper mathematical syntax when you enter equations. The required syntax is similar to that used when entering mathematical equations in computer code or spreadsheet formulas. For example, say you wanted to change the equation shown above to have the term in brackets multiplied by the constant $\mathbf{Y_a}$. The following syntax would not be correct, because the multiplication operator is missing:



Incorrect multiplication syntax

The equation shown below, with the correct multiplication syntax would be acceptable:



Correct multiplication syntax

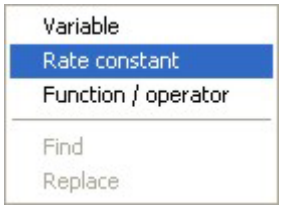
Equation Editor Text Editing Features

The equation editor has features commonly found in text editors. You can highlight a section of equation text by dragging your mouse cursor over it. You can then move the highlighted selection by clicking and dragging the mouse cursor. The following keyboard shortcuts for using the Windows clipboard also are available:

- To cut text, use the **Ctrl+x** keyboard combination;
- To copy text, use the **Ctrl+c** keyboard combination;
- To paste text, use the **Ctrl+v** keyboard combination.

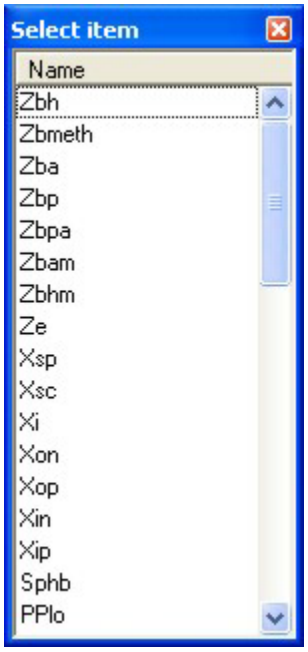
Equation Editor Popup Menu

The equation editor also offers the powerful functionality of a right-click popup menu that makes writing your equations much easier. If you right-click your mouse button anywhere within the equation editor window, you will be presented with a popup menu similar to the one shown below:



Equation editor right-click popup menu

Selecting one of the popup menu choices will open a small window containing a list of variables, constants, or function/operator templates that you can place into your equation. For example, selecting the **Variable** option opens a window that lists the BioWin state variables, shown below:

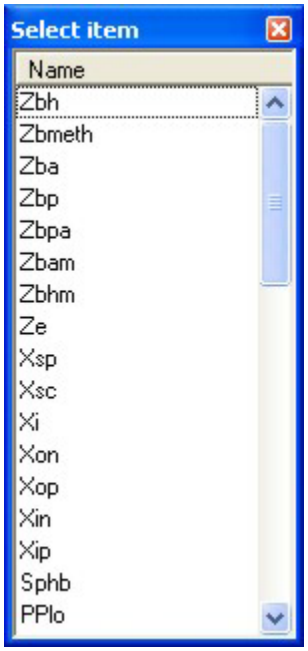


State variable selection window

To place a variable from this list into your current equation, simply locate the variable in the list and double-click it. The popup window will close, and you will return to your equation, where you will see that the variable has been placed where you had located your cursor. Two important points should be mentioned regarding the use of these popup windows:

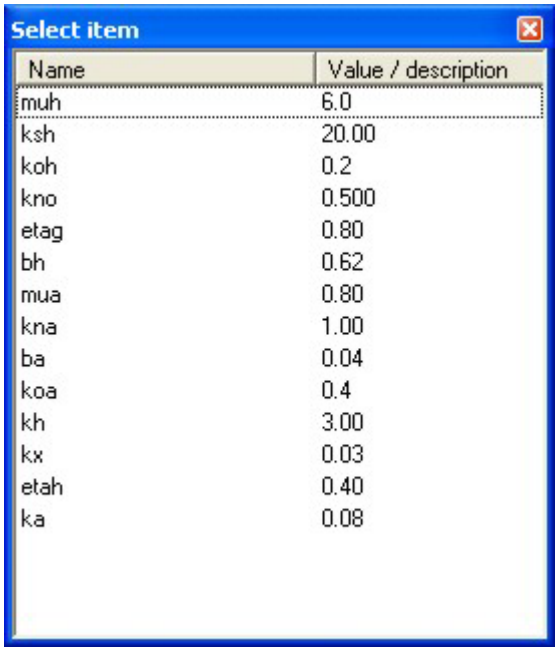
1. To close a popup window without adding an item to your equation, click the small **x** located in the upper right corner of the popup window.
2. When you double-click an item and add it to your equation, BioWin also adds the item to the Windows clipboard. Pressing **Ctrl+v** will add the item to the equation, until another item is added to the clipboard.

Selecting the **Stoichiometry constant** option opens a window that lists the stoichiometry constants that you have defined for the model that you currently are working on. For example, in the picture shown below, note that the constants listed coincide with those shown in the **Stoichiometric constants** list in the main model builder window shown previously. To place a constant from this list into your current equation, simply locate the constant in the list and double-click it. The popup window will close, and you will return to your equation, where you will see that the constant has been placed where you had located your cursor.



Stoichiometry constant selection window

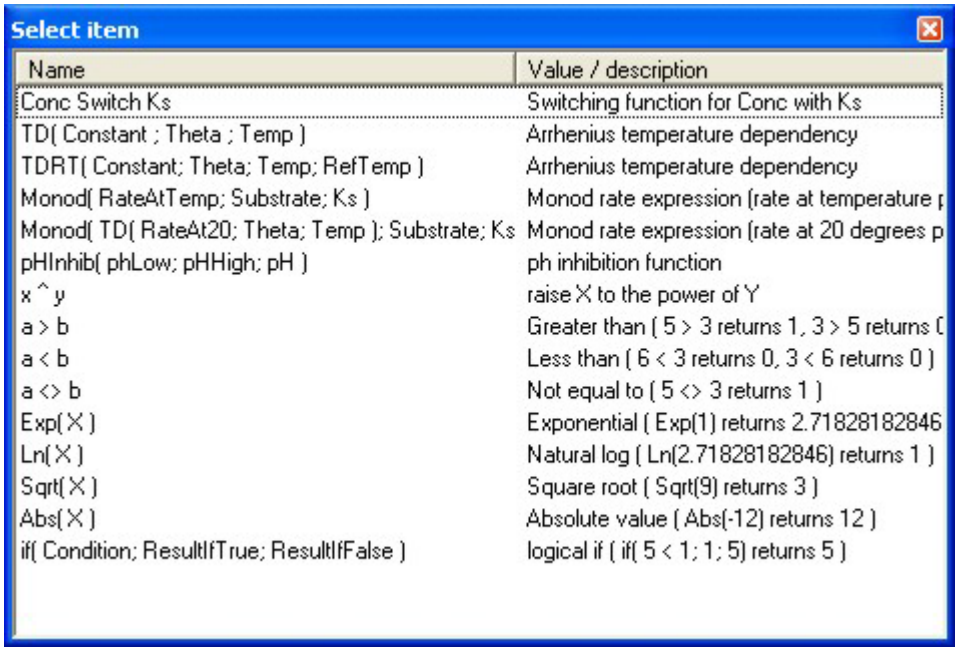
Selecting the **Rate constant** option opens a window that lists the rate constants that you have defined for the model that you currently are working on. For example, in the picture shown below, note that the constants listed coincide with those shown in the **Rate constants** list in the main model builder window shown previously. To place a constant from this list into your current equation, simply locate the constant in the list and double-click it. The popup window will close, and you will return to your equation, where you will see that the constant has been placed where you had located your cursor.



Rate constant selection window

Selecting the **Function / operator** option opens a window that lists a number of function / operator templates that you may place in your equation. To place a

function or operator from this list into your current equation, simply locate the function or operator in the list and double-click it. The popup window will close, and you will return to your equation, where you will see that the function or operator template has been placed where you had located your cursor.



Function / operator selection window

For example, suppose you selected the **x^y** template from the list. Using our previous DO stoichiometry equation example, you would see the following:

Inserting a function template into an equation

You could then quickly replace the **x** and **y** placeholders by highlighting them, right-clicking, selecting **Variable** or one of the constants options, and inserting an item from a popup window. Functions and operators are described with examples in the next section.

Available Functions and Operators

A more comprehensive list of functions and operators is tabulated below:

Operator	Description	Example
^	power operator	2 ^ 3 returns 8
*	multiply operator	3 * 4 returns 12
/	divide operator	8 / 4 returns 2
\	module operator. Returns the module of a division.	5 \ 2 returns 1
+	sum operator	3 + 7 returns 10

-	subtract operator	10 - 4 returns 6
>	greater than operator	4 > 3 returns 1 (true), and 1 > 6 returns 0 (false)
<	less than operator	6 < 3 returns 0 (false), and 2 < 9 returns 1 (true)
>=	greater than or equal to operator	6 >= 3 returns 1 (true)
<=	less than or equal to operator	7 <= 3 returns 0 (false)
<>	not equal to operator	5 <> 3 returns 1 (true)
=	equal to operator	3 = 8 returns 0 (false)
if	logical if operator. Follows format if(condition; result if true; result if false)	if(5<1; 1; 5) returns 5 since the condition tests false
and	logical and operator	3 and 2 returns 1 (true, because both are true)
or	logical or operator	3 or 0 returns 1 (true)
xor	logical xor operator	1 xor 1 returns 0 (false)

Constant	Value
pi	3.1415926535897932385
e	2.71828182846 - Same as Exp(1)

Function	Description	Example
Conc Switch Ks	Applies the Ks switch to the variable Conc	**Alk Switch KsAlk** applies the switching function **KsAlk** to the variable **Alk**
TD(Const;Theta;Temp)	Applies an Arrhenius temperature dependency relationship using the supplied arguments	$Const \cdot \theta^{Temp-20}$
TDRT(Const;Theta;Temp; RefTemp)	Applies an Arrhenius temperature dependency relationship using the supplied arguments	$Const \cdot \theta^{Temp-RefTemp}$
Monod(RateAtTemp; Substrate; Ks)	Places the supplied arguments into a Monod function	$RateAtTemp \cdot \frac{Substrat}{K_S + Subst}$
Monod[TD(Const;Theta; Temp); Substrate; Ks]	Places the supplied arguments into a temperature-dependant Monod	

	function	
PhInhib(phLow; pHHigh; pH)	Places the supplied arguments into a two-sided pH inhibition function	$\frac{K_{LpH}}{K_{LpH}+pH} \bullet \frac{pH}{K_{HpH}+pH} \bullet \frac{K_{LpH}+\left(\frac{K_{LpH}+K_{HpH}}{2}\right)}{K_{LpH,HET}} \bullet \frac{K_{HpH}+\left($
Neg	Returns the negative of argument	Neg(5) returns –5
Not	Returns 1 (true) if argument is 0 (false). Returns 0 (false) if argument is not equal to 0	Not(1) returns 0
Re	Returns the real part of a complex number	Re(5 + 4j) returns 5
Im	Returns the imaginary part of a complex number	Im(5 + 4j) returns 4
Exp	Returns the exponential of argument	Exp(1) returns 2.71828182846
Sin	Returns the sine of argument in radians	Sin(pi/2) returns 1
Cos	Returns the cosine of argument in radians	Cos(pi/2) returns 0
Tan	Returns the tangent of argument in radians	Tan(pi/4) returns 1
Asin	Returns in radians the inverse sin of argument	Asin(1) returns 1.57079632679 (pi/2)
Acos	Returns in radians the inverse cosine of argument	Acos(1) returns 0
Atan	Returns the inverse tangent of argument	Atan(1) returns 0.785398163397
Abs	Returns the absolute value of argument (or the complex module)	Abs(-5) returns 5
Sqrt	Returns the square root of argument	Sqrt(16) returns 4

U	Returns the step-function of argument	U(-5) returns 0, and U(3) returns 1
Rect	Returns the rect function of argument	If absolute value of argument is less the 0.5, returns 1. Otherwise, returns 0
sinc	Returns the sinc function of argument	
Ln	Returns the natural log of argument	Ln(2.71828182846) returns 1
Round	Returns the rounded number of argument	Round(4.8) returns 5
int	Returns the int part of argument	int(3.2) returns 3
Odd	Returns 1 (true) if rounded argument is odd. Returns 0 (false) if rounded argument is even.	Odd(3) returns 1
Rand	Returns a random number between 0 and argument	Rand(3) returns a random number between 0 and 3
Sinh	Returns the hyperbolic sine of the argument	Sinh(1) returns 1.17520119364
Cosh	Returns the hyperbolic cosine of the argument	Cosh(1) returns 1.54308063482
Tanh	Returns the hyperbolic tangent of the argument	Tanh(1) returns 0.761594155956
Diode	Returns 0 if the argument is less than or equal to 0. Returns the argument if argument is greater than 0	Diode(-6) returns 0 and Diode(3) returns 3

Verifying Equations

As you enter equations in your model, you may want to use the **Verify equations…** button to ensure that you have no mistakes in your equations. When you click this button, BioWin checks your equations for undeclared variables and constants (rate and stoichiometric). BioWin also checks your equations for the proper syntax. Finally, BioWin attempts to calculate the model equations. This may be useful in discovering equation formula errors such as division by zero.

You may find it useful to use the **Verify equations…** button each time you finish entering an equation.

Copying, Pasting, and Printing Equations

A number of options for manipulating your model equation entries are available by right clicking in the stoichiometry and rate equation matrices.

If you right-click and select **Copy** from the resulting popup menu, the contents of the stoichiometry or rate matrix (including column headings, the values in non-editable cells, and the values in editable cells) will be copied to the clipboard. The clipboard contents then may be pasted into another application such as a word processor (all values will be separated by tabs) or a spreadsheet (each value will be placed in a separate cell).

You also may paste data from the clipboard into the stoichiometry or rate matrix by right-clicking and selecting **Paste** from the resulting popup menu.

Note : The cell that currently is selected in the stoichiometry or rate matrix corresponds to the first value in the first column of the data that you are pasting from the clipboard.

If BioWin does not recognize the format of the data that you are attempting to paste in, or if you select **Paste special** or **Paste from file**, the import wizard will be opened.

If you want to print the stoichiometry or rate equation matrix in tabular form, then you can right-click and select **Print** from the resulting popup menu. When you do so, you will be presented with the BioWin print dialog box, which is explained in detail in the **Model Parameter Editor** section earlier in this chapter.

Managing Models

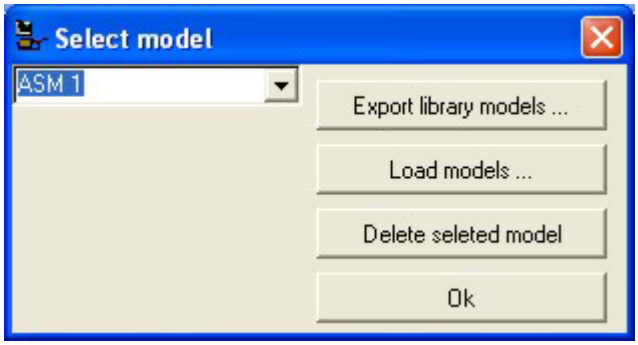
Models that you create can be either project-specific (i.e. they are available only within the project that they were created), or available to all of your BioWin projects.

If you create a model using the **Project|Model Builder…** command, then that model will be used by all model builder elements that you place in your configuration. You can also have individual models that are local to their model builder elements (when you select the **Local Builder model** option on the **Operation** tab).

Model Library

You can make a model available to all of your BioWin projects by adding it to the **Model Library**. When you have finished inputting your model in the model builder, you can add the model to the model library by clicking the **Add to model library** button. Before you do this, be sure to enter a descriptive name in the **Model Name** text edit area for your reference purposes.

Once you have added a model to the model library, it will be available to all of your BioWin projects. You can open any model in the library into the model builder by clicking the **Select from model library…** button. Doing so will open the following dialog box:



The Model Library management dialog box

You may use the drop list selector to choose the model that you wish to load into the model builder, and press **OK** to do so. You may also remove a model from the current library by selecting it and then clicking the **Delete selected model** button.

You may also save a particular model library to disk as a further management method. For example, say that your current library consists of activated sludge models. You may want to begin working on digester models in the model builder, but you want to keep these separate from the activated sludge models. To achieve this end, you could use the **Export library models…** button, which opens a **Save model library** dialog box. This dialog box operates the same as a standard **File Save** dialog box, and allows you to save the current model library in the BioWin ***.GML** format. In our example, you may want to save the activated sludge models as **ActivatedSludgeModels.GML**. When you are finished with your digester models, you may want to save that library as **DigesterModels.GML**.

You can load a saved **GML** file into the current library in one of two ways. When you click the **Load models…** button, you are asked whether you want to **Add models from file to this library (press "Yes"), or replace this library (press "No")**. Selecting either option will present you with an **Open model library** dialog box that operates exactly like a standard **File Open** dialog. However, choosing the first option will append the models from the loaded **GML** file to the model list in the current library. Choosing the second option will make the models from the loaded **GML** file the only models in the current library.

Importing and Exporting Models

You can also manage individual models. You can save the model you are currently editing to disk by clicking the **Export to file…** button. This opens the **Save model file** dialog box, which operates exactly like the standard **File Save** dialog box. You may save your model in one of three possible file formats:

1. **BioWin Model files** (*.mod)
2. **Text files** (tab delimited) [*.txt]
3. **CSV files** (no spaces) [*.CSV]

BioWin Model files (*.mod) are an excellent method for saving your files as they move to and from BioWin seamlessly, and they also can be imported into Microsoft Excel as comma delimited files.

You may also load a model from a file using the **Import model from file…** button. This will open the **Open model file** dialog box, which operates exactly like a standard **File Open** dialog. You may open a file saved in any of the three formats discussed above.

BioWin Tutorials and Examples

Learning Objectives

This set of tutorials and case studies is designed as a training exercise in the application of BioWin. The primary objective is to provide "how to…" training on using the BioWin software itself. The case studies are not intended as a course in wastewater treatment process engineering. Nevertheless, several of the case studies focus on process applications and identify interesting design and operating issues.

Start off with Tutorial #1. This explains the basic concepts for applying BioWin and provides an overview of features. You will be viewing a previously created file from the installation **Data** directory. The remaining tutorials all involve setting up new systems. Each tutorial is broken up into a number of subsections.

The tutorial examples ask you to save files with the name format **My Tutorial XX** in the **Tutorials** subdirectory of the **Data** directory. Completed tutorial configurations are also stored in that directory for reference as **Tutorial XX**.

Suggestion: Keep this Help file open and follow each tutorial step-by-step, switching back and forth to BioWin. Alternatively, print a copy of the tutorial chapter (for printing instructions see **How to print the manual** in the "*Welcome To BioWin*" chapter).

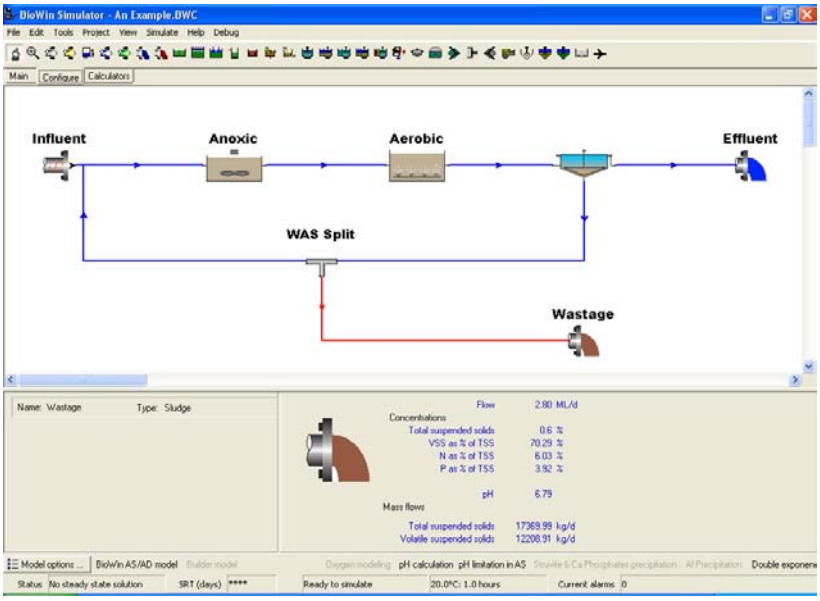
TUTORIAL 1 - BioWin Familiarization

This familiarization tutorial demonstrates a number of the features in BioWin. Aspects covered in this tutorial include the basic BioWin interface, loading a BioWin configuration file, specifying data for configuration elements, and running steady state and dynamic simulations.

The interface and loading a file

Start BioWin and view the main simulator window. All simulation tasks are executed from here. The interface consists of menus, toolbars, the drawing board, summary panes and a status bar. For a detailed description, view the "*Main Simulator Window*" chapter. In this tutorial you will only get a brief overview.

1. From the **File** menu, click on **Open** and load the **An Example** configuration file from the **DATA** directory. The screen view will be similar to that shown below.



The example configuration

Element Selection Tool

2. Move the cursor over the toolbar. A fly-by hint appears when you pause over a button. The **Select/Drag/Edit** tool at the left of the lower tool bar should be depressed (i.e. on) at this stage.
3. A status bar at the bottom of the window displays various pieces of information.

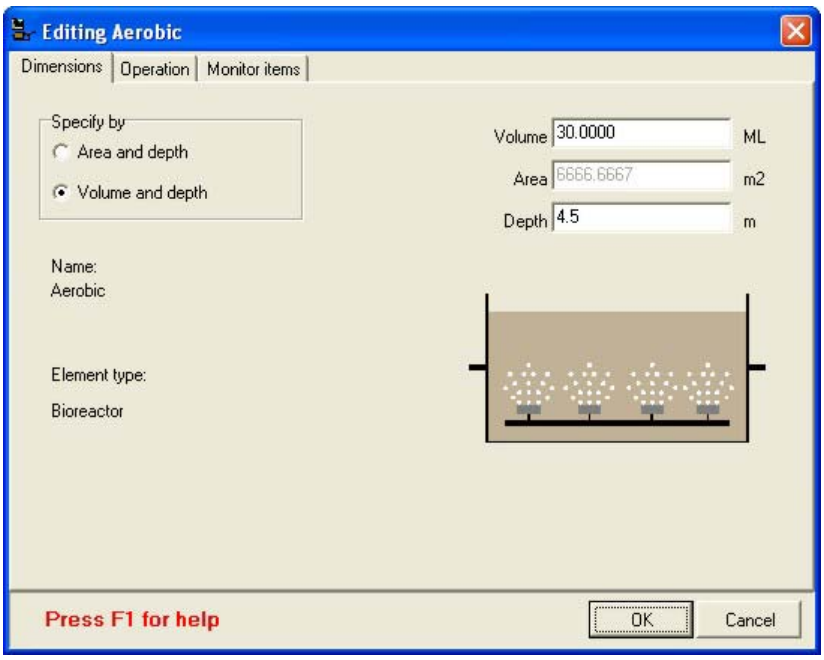
Selection Cursor

4. Move the arrow cursor across the drawing board. The cursor changes to a hand as you cross over elements on the drawing board. When you pause over an element, information on that element appears in the two panes below the drawing board – physical data in the left pane and performance data in the right pane. This function allows you to get a summary overview of system information.
5. Move the cursor over an element and click the right mouse button. A local menu appears. [Don't select any options yet!]. This will allow you to access various options for that element (see below).

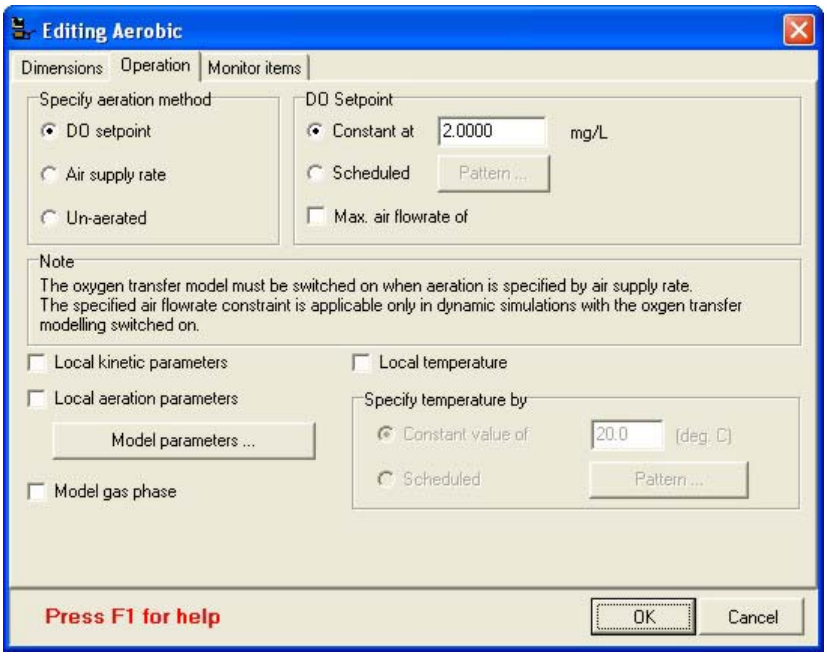
Hint: As a general rule when using BioWin, clicking the right mouse button often helps!

Physical and operational data

60. Move the cursor over the **Aerobic** element (a completely mixed aerated bioreactor) and double click – or click the right mouse button on the element and select the **Properties** command. A tabbed editing dialog box opens (see below). This contains all the physical and operational data for the element. View the information on the **Dimensions** and **Operation** tabs. [Don't change any information yet. We'll accept the sizing of this aerated zone with DO controlled at a setpoint of 2 mg/L].



The Bioreactor Dimensions tab



The Bioreactor Operation tab

61. Now try double-clicking on other elements and view the details for this configuration.
62. Click on an element and keep the left mouse button depressed. You can drag and drop the element in a new position, and re-arrange the configuration to your liking.

Hint: Try right-clicking on the arrow head of a pipe, and view the **Properties**. There are a number of options for re-arranging the pipe layout.

Checking influent data

1. Double click on the **Influent** element, and click on the **Edit data** button. At this stage we won't change any data.

Hint: When viewing influent data, point the cursor at a column heading and click the right mouse button. There are many options for entering and manipulating data.

2. **Close** the dialog.

Hint: The pane at the lower right displays the flow-weighted average influent concentrations.

Viewing information and simulation results

The panes below the drawing board provide a limited overview of system information. Comprehensive information can be viewed in two ways – via the **Explorer** or in the **Album**.

The Explorer button

63. Select Explorer from the View menu – or click on the Explorer toolbar button – or press Ctrl + E. This opens the Explorer - a tree-like view of system information.
64. Experiment with expanding different levels. Hint: Try double clicking on an element name or the parameters item in the right panel.

The Album button

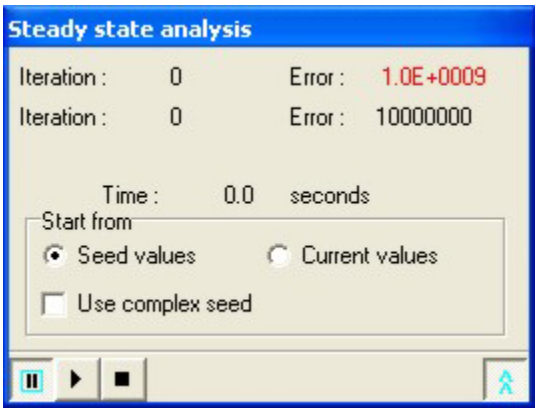
65. Select Album from the View menu – or click on the Album toolbar button – or press Ctrl + A. This opens the Album. This contains user-customized information in the form of custom tables, pre-formatted element information, and charts. The Album can contain many pages of information.
66. Click on the page name tabs along the bottom and view the different examples. Hint: Try clicking the right mouse button on different parts of the Album pages (including the name tabs at the bottom).
67. Select Add page from the Album menu, and choose one of the layout options. After adding a page, click the right mouse button on a blank panel and experiment! Expect to encounter difficulties at this stage – subsequent tutorials will provide detailed instructions.

Running a steady state simulation

Steady state simulations provide a solution for the system based on the flow-weighted average influent loading to the system (and the time-weighted average for any timed operational changes such as a schedule of DO setpoints in an aerated reactor).

Steady state button

68. Select the Steady state command in the **Simulate** menu – or click on the **Steady State** button on the toolbar. This opens the simulator player dialog. **Hint**: If you re-position the simulator player dialog at a convenient spot on the screen that's where it'll appear next time.
69. Click on the **play** button. A dialog box appears when BioWin has found the solution.



The steady state solver dialog box

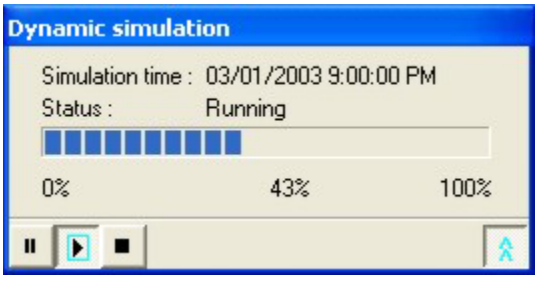
Note: Most steady state solutions are found in ten or so iterations. In unusual circumstances the solver may "stick" – that is, the error value does not change from iteration to iteration. In this situation click on the stop button. Often this indicates a difficulty with the influent data such as a nutrient deficiency (or an Alkalinity deficiency in an aerobic digester, perhaps). Alternatively, you may have a "difficult-to-solve" system. One trick is to try conservative solver settings. To do this, select the menu command **Project|Current Project Options**, click the **Numerical Parameters** tab, and click the **Options**… button in the **Steady State Solver** group. At the bottom of the resulting dialog box there is a large button that you can click to set conservative solver parameters (see **Steady State Solver Options** in the **Manage Projects** chapter).

Running a dynamic simulation

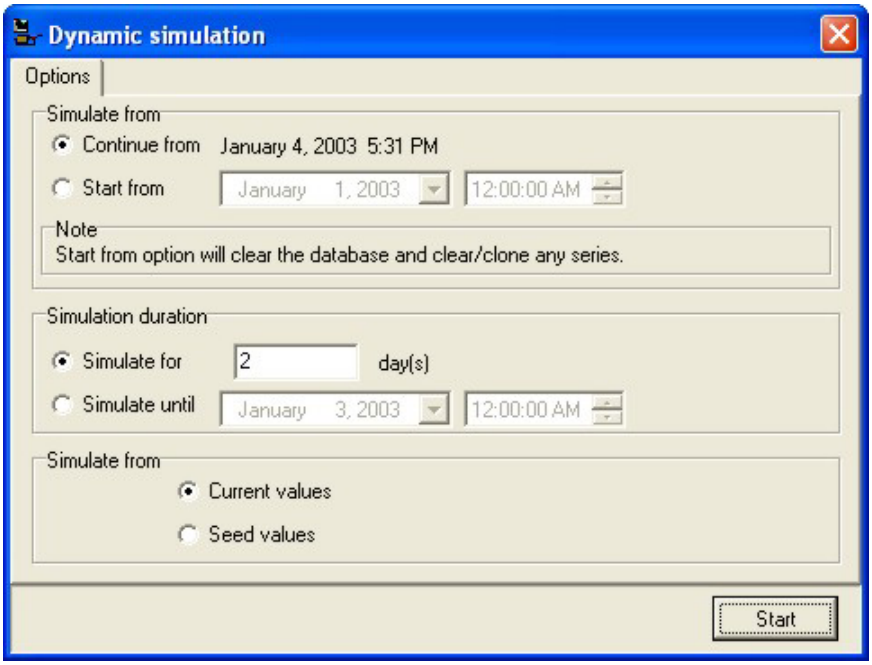
Dynamic simulations show the time-varying system response based on the time-varying influent loading to the system (and subject to any time-varying operational changes such as a schedule of DO setpoints in an aerated reactor).

Dynamic Simulation button

70. Select the Dynamic simulation command in the **Simulate** menu – or click on the **Dynamic Simulation** button on the toolbar. This opens the simulator player dialog.
71. Click on the play button. This brings up a dialog box where you can set various options such as the duration of the simulation. Clicking on the **Start** button starts the dynamic simulation.



The dynamic simulator control dialog box



The dynamic simulation options dialog box

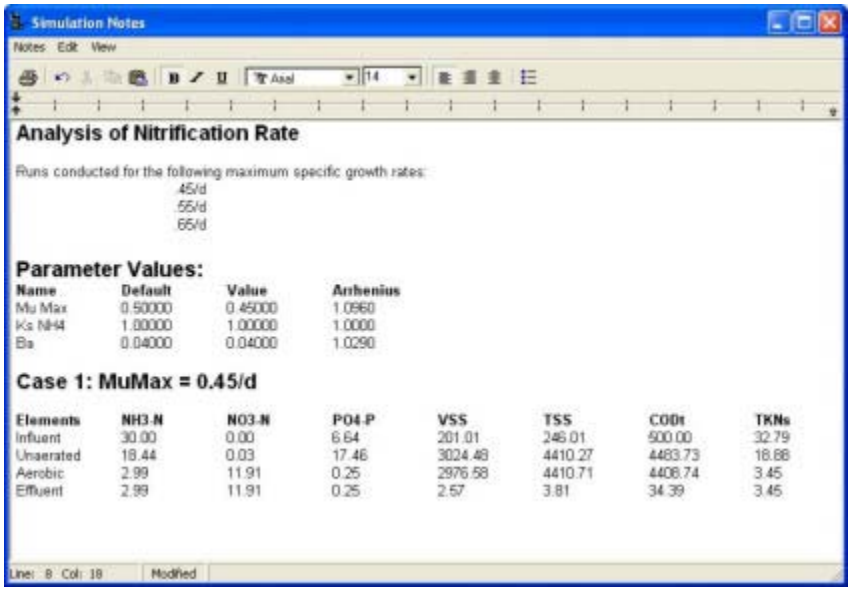
Note: Even if you are only interested in dynamic system response, it is useful to first calculate the steady state solution, and then start the dynamic simulation from these "Current" values.

Note: In the **Album** a time-series chart set up for 24 hours may appear blank or may not reflect a change you expected to see. Perhaps you need to change the scale on the bottom axis depending on what you specified as the starting date for the simulation.

Keeping track of things and generating reports

Simulation Notes button

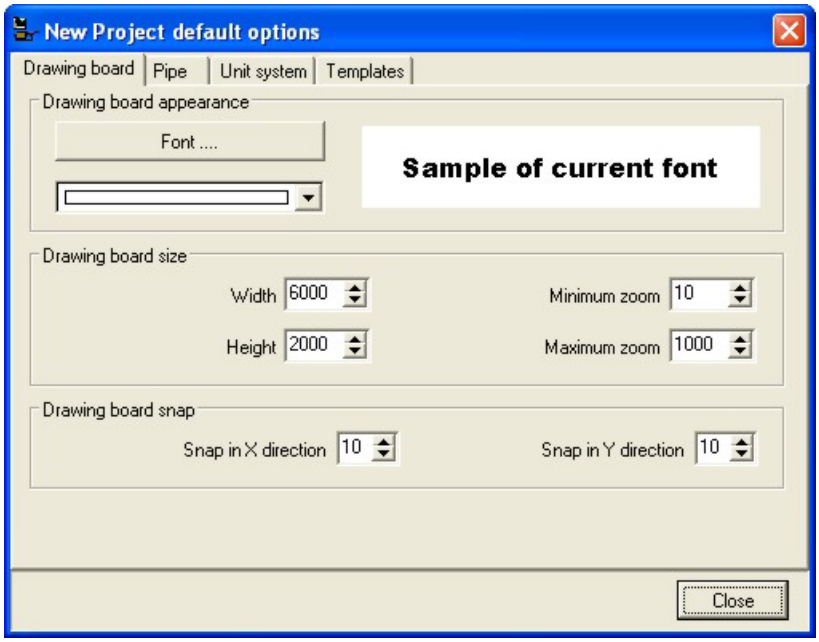
BioWin offers a number of features to aid in creating attractive, professional reports. You have your own internal Notes editor (shown below) to help keep track of project details. Access this scratch pad from the **Project|Notes** command or by clicking on the Notes button on the toolbar. It also is very easy to get results from BioWin into your word processor, spreadsheet or other applications. Charts, tables, the drawing board view of the system layout, etc. can be copied from BioWin and pasted to your report. Tables can be exported as tabbed text and then quickly converted to tables.



BioWin's internal notes editor

Customizing BioWin

There are a number of parts and features of BioWin that can be customized to look how you want. When you customize BioWin, you essentially are changing the default settings of the BioWin work environment and all new projects that are created therein. This functionality is accessed via the **Project|New Project Options** menu command used to customize BioWin. A detailed description of the features accessed from this tabbed dialog is provided in the "*Customizing BioWin*" chapter.



Many facets of BioWin may be customized to suit your needs

The **Tools|Customize** command defines your "default" setup for when you start a new project; for example, you may always want to start with US units. You can

override these preferences for the current project through the **Project|Current Project Options** command.

Since project options are file specific, they "travel" with that file. For example, if you define a set of project options for "Project A" on your copy of BioWin and then open the "Project A" file in someone else's copy of BioWin, you still will see your defined project options. As before, these project options will override any similar settings that the owner of the other copy of BioWin has set as defaults using the **Tools|Customize** command. For more information on the various project options that may be set, please see *Managing BioWin Projects*"

TUTORIAL 2A - Building a Configuration

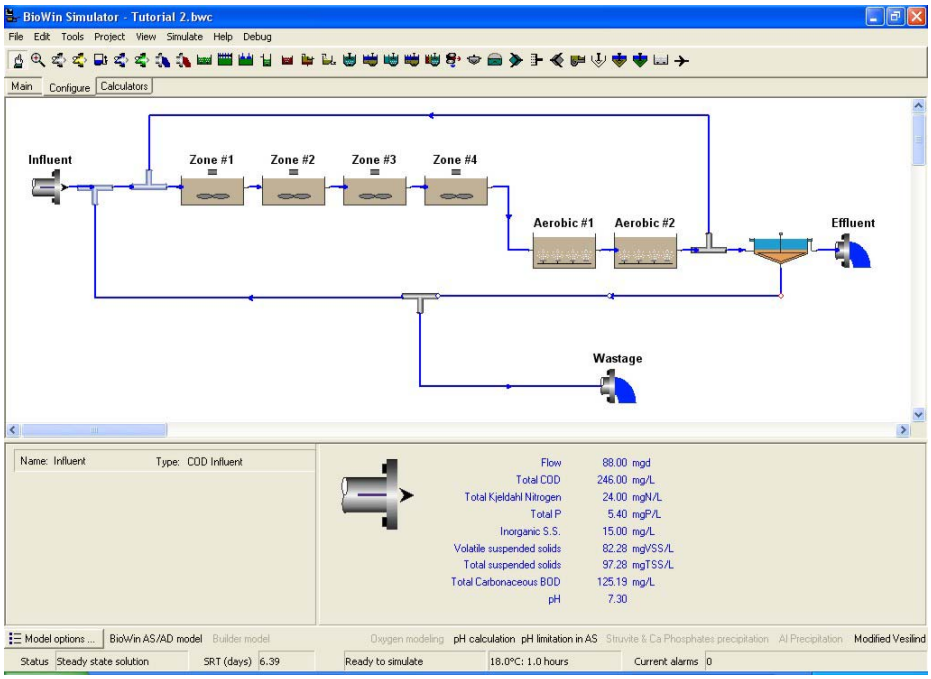
This tutorial demonstrates how to build a new configuration and add tables to the Album. Aspects covered in this tutorial include building a BioWin configuration, moving and connecting elements on the drawing board, specifying data for elements, changing model parameter values, and setting up tables to record your simulation data.

The tutorial 2A system

A city in the U.S.A. has a nitrification/denitrification system – a Modified Ludzack Ettinger configuration. Phosphorus removal is achieved in a tertiary chemical precipitation system. The client experiences problems in the tertiary system. You want to investigate achieving P removal biologically in the existing tankage. The system has the following characteristics:

Unaerated reactors:	Four (each 1 MG)	Depth = 12 ft	DO = 2 mg/L
Aerated reactors:	Two (each 7.5 MG)	Depth = 12 ft	DO = 2 mg/L
Clarifier (Ideal):	Area = 123,000 ft^2	Depth = 14 ft	
Influent:	Average Flow = 88 MGD		
	COD = 246 mg/L	TKN = 24 mgN/L	
	TP = 5.4 mgP/L	ISS = 15 mg/L	
	Alkalinity = 6 mmol/L		
Wastewater fractions:	f_{BS} = 0.12	f_{UP} = 0.10	
	f_{US} = 0.07	f_{NA} = 0.75	
RAS recycle:	44 MGD (50%)		
NML recycle:	264 MGD (300%)		
Wastage rate:	1 MGD (constant rate)		
Temperature:	18°C		

Nitrification rate:	0.8 /d		



The Tutorial 2A system configuration

Adding elements to the drawing board

Note: When building the configuration, don't interchange mixers with splitters or influents with effluents.

Note: In this tutorial we are using an ideal secondary settler. **Tutorial 1** used a model settler.

Element Placement Cursor

72. Run BioWin and change to US units via the **Project|Current Project Options** command.
73. Add each of the units shown in the screen view above. [We'll connect units with pipes later]. Repeat the following three steps as you build the system in the drawing board:·

- Click the button corresponding to the element you want on the configuration toolbar.·
- Move the cursor onto the drawing board. When you do this, the cursor will change to the element placement cursor. Click on the drawing board where you want the element to be placed.·
- Change the names of the elements from the defaults to those shown in the screen view above (i.e. Influent, Zone #1, Zone #2,.Zone #3, Zone #4, Aerobic #1, Aerobic #2, Settler, Effluent, Wastage). Right-click on each element and select the **Name**… command from the popup menu.

Note: No names appear for the mixers, splitters and the settler in the screen view above. This is one of the customizable features of BioWin. You can make your own selection from the **General** tab via the **Tools|Customize** command.

Note: Your configuration may extend beyond the visible drawing board view. You may wish to change the drawing board scale from the drop down list on the toolbar.

Note: This configuration includes mixers for the RAS stream and mixed liquor recycle in front of the first bioreactor. It is not necessary to include these mixers – the streams could be connected directly to the front of the bioreactor.

Rearranging and moving elements on the drawing board

If you want to change an element's position:


Element Selection Tool

74. Click on the element selection tool from the Configure toolbar.
75. Move the cursor over the element on the drawing board you wish to move.
76. Click on the element and while holding down the mouse button, drag the element to the desired new location.

Note: You also can move multiple elements simultaneously. Select the group of elements you wish to move, click on one of them, and drag the group to the desired new location.

If you want to change an element's vertical or horizontal orientation:

1. Click on the element selection tool from the configuration toolbar.
2. Right-click the element, and from the resulting popup menu, choose **Flip horizontal** or **Flip vertical** (the latter option only is available for elements such as splitters and mixers).

Connecting elements with pipes


Pipe Tool


Pipe Start Tool

77. Click the pipe tool on the **Configure** toolbar.
78. When you move the cursor onto the drawing board, the cursor will change to the "start" cursor.
79. Place the cursor over the element area where you wish the pipe to start

from.

Pipe Start "OK"

80. If the location is appropriate, a set of crosshairs will appear on the "pipe start" cursor.

No Pipe Allowed

81. If the location is inappropriate, the cursor will change to a circle with a slash through it to indicate that a pipe may not begin at that location.
82. Click the left mouse button once and move the cursor to the desired location of the element where you wish the pipe to end and click the left mouse button again.

Pipe End Cursor

83. As you move the pipe towards the element where you wish it to end, the cursor will change to the "pipe end" cursor.

No Pipe Allowed

84. If the location of the pipe terminus is appropriate, this cursor will remain.
85. If the location is inappropriate, the cursor will change to a circle with a slash through it to indicate that a pipe may not end at that location.
86. **Repeat steps 3-9** until you have connected all your elements with pipes.

This is important for arranging the configuration layout.

Note: To re-arrange a pipe's position, click once on the arrow head of that pipe. A series of circles appear at points along the pipe. Try dragging-and-dropping.

Note: Try right clicking on the arrow head of a pipe, and view the **Properties**. There are a number of options for re-arranging the pipe layout and selecting pipe style.

Specifying physical and operational data

We now wish to enter all of the physical and operational data for this system specified earlier. Each element (except influents and effluents) requires physical data, and this is specified in terms of either **Volume/Depth** or **Area/Depth**. Operational data depends on the type of element. For example, units such as bioreactors require information on aeration and DO levels. Units such as splitters and settlers also require information on the flow split in the side stream or underflow.

Note: There are many options for specifying operational data for the units in BioWin. We only touch on a few options in this tutorial. More complete information on the different options for each unit type is provided in the "*Element Types*" chapter.

To specify data for each element (leave the influent for now):

1. Double click on the element – or click the right mouse button and select the **Properties** command. Specify data from the information listed earlier.
2. For the influent element specify the type as **Constant**. In this tutorial we only consider steady state performance.

File Save button

When you are finished use the **File|Save As…** command – or click on the Save button on the toolbar - to save the configuration as **My Tutorial 2** in the **Data/Tutorials** directory.

Note: Mixers and splitters can be defined as "dimensionless"; that is, nodes without volume. This is preferable to using very small volumes compared to other process units in the configuration because small volumes may result in slow dynamic simulations.

Specifying process temperature(s)

Specify the global temperature for the system (18°C) via the **Project|Temperature…** menu.

Note: You can specify a local temperature for many of the units. For example, view the **Operation** tab in the bioreactor dialog.

Changing model parameters

In this example we must specify a maximum nitrifier growth rate (referenced to 20°C) of 0.80 /day. All model parameters for the project are viewed/changed via the **Project|Parameters…** command. In this case we want to go to the **Kinetic** menu and change the autotroph MuMax from the default of 0.90 to 0.80 /day.

Note: You can specify local model parameters for many of the units. For example, view the **Operation** tab in the bioreactor dialog and note the check box labeled **Local kinetic parameters**.

Checking that all data have been specified

Before running a simulation BioWin must check to see that data have been specified for each of the elements.

Check Simulate Data button

1. Select the **Check simulate data** command from **Simulate** menu – or click on the Check data button on the toolbar. A dialog box appears with a list of elements for which data have not been specified and/or elements without pipe connections.

Note: Don't worry if you forget the check. BioWin will remind you about missing data. It may seem unnecessary to check data for elements (such as mixers) where you generally will accept the default values. This only is necessary once.

Adding tables to the Album

You've now completed setting up the configuration. [Remember: You've yet to run a simulation so data values may be garbage!]. We now want to add data tables to the Album. Let's set up a table similar to that shown below on Page 1 of the Album. This has rows for influent, all bioreactors, effluent and wastage, and columns for NH3-N, NO3-N, PO4-P, ISS, flow, VSS and TSS.

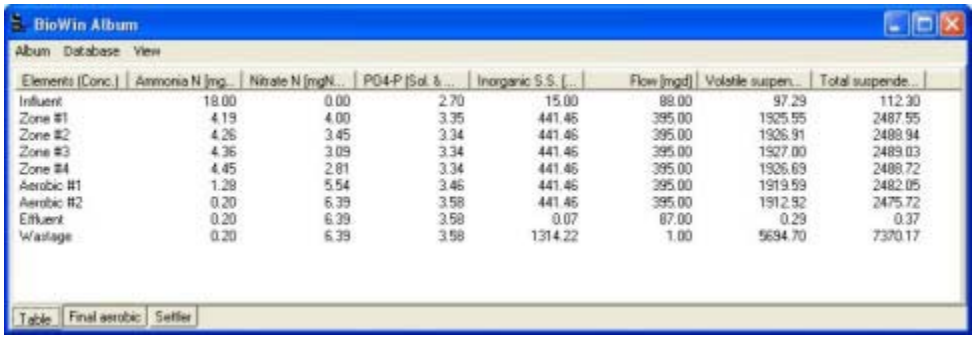
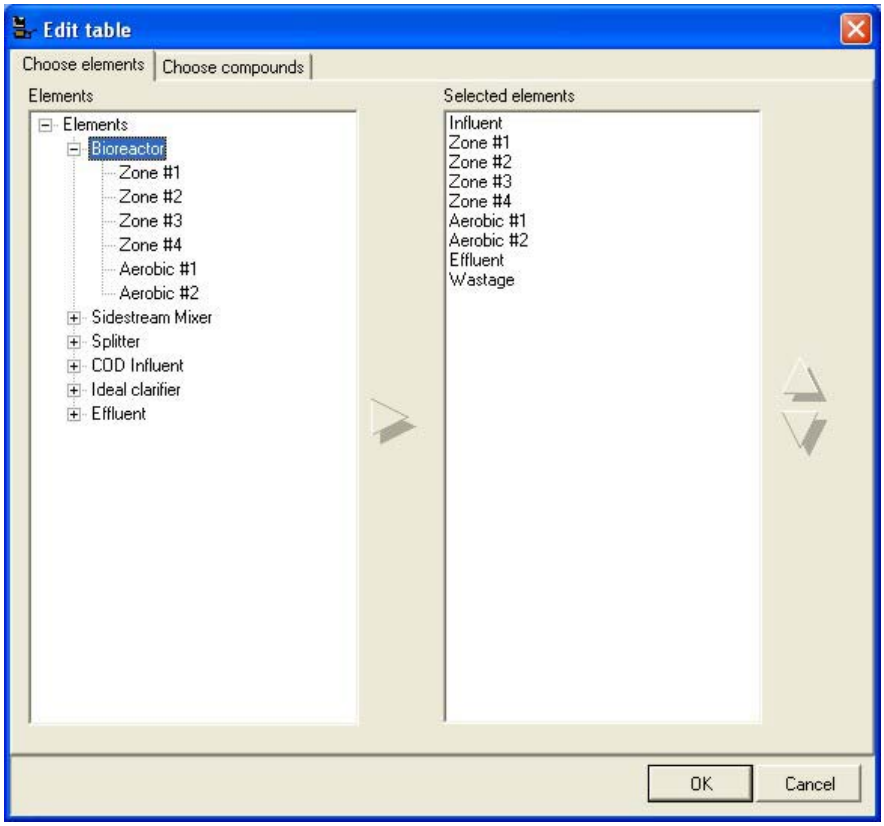


Table from the tutorial configuration

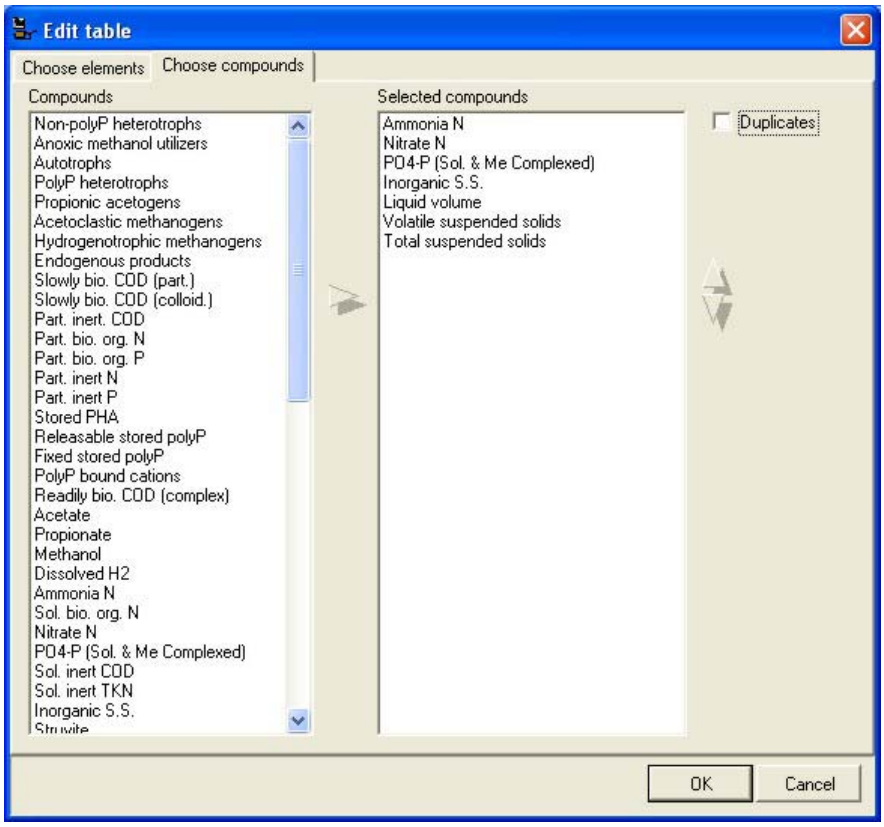
Album button

1. Select **Album** from the **View** menu – or click on the **Album** toolbar button – or press **Ctrl + A**. This opens the **Album** – it's blank for now.
2. Select **Add Page** from the **Album** menu and click **OK**.
3. Right-click on the album page.
4. Select **Table** from the popup menu.



Dialog box used for choosing elements to include in your table

5. A dialog will open with the **Choose elements** tab displayed. From the **Elements** tree view, select the element(s) that you wish to include in the table.
6. If you wish to include all the elements from a group (for example, all the bioreactors), click on the group heading and then click the right-pointing arrow to move them all to the **Selected Elements** list.
7. If you wish to include only certain elements from a group (or groups), then click on the plus sign (+) next to the group heading to expand it, click on the specific element you want to include, and then click the right-pointing arrow to move it to the **Selected Elements** list.
8. If you want to change the order in which the **Selected Elements** will appear in the table, move the elements around by clicking on them and then clicking the **Up/Down** arrows.
9. Now click the **Choose compounds** tab. In the **Compounds** list, select the compounds that you wish to appear in your table, and add them to the **Selected compounds** list by clicking the right-pointing arrow. To select multiple compounds:
10. Select contiguous multiple compounds by clicking on the first desired compound, holding down the **Shift** key, and clicking the last desired compound.
    - Select non-contiguous multiple series by holding the **Ctrl** key and clicking on the different desired compounds.



Dialog box used for selecting compounds for inclusion in your table

11. If you wish to re-add certain compounds, place a check in the box labeled **Duplicates**, and re-add the compounds.
12. Click **OK** to finish.

Note: You can change the order of rows and columns in the table very easily. Right-click on the table, select the **Edit table** command, and use the Up/Down arrows.

Note: Try clicking the right mouse button on different parts of the **Album** pages (including the name tabs at the bottom – you can change the name of your tab to "Table" from "Page 1" if you wish).

Note: Moving the cursor over elements on the drawing board gives you a sneak preview of data in the panes below the drawing board.

Adding element information to the Album

The previous section showed how to set up a table in the Album. You can also add pre-formatted element-specific information to the Album. Let's add information on the last aerated bioreactor (Aerobic #2) to Page 2 and for the settler to a new page of the Album.

We'll do this by two different methods.

 *Album button*

1. Select **Album** from the **View** menu – or click on the **Album** toolbar button – or press Ctrl + A.
2. Select **Add Page** from the **Album** menu and click **OK**.
3. Right-click on the album page.
4. Select **Element info** from the popup menu.
5. Select **Aerobic #2** from the drop-down element list, and click on the **Summary view** radio button.

We'll add a similar table to the Album for the settler, but using a different method.

1. Close the **Album**, and move the cursor over the settler in the drawing board.
2. Right-click and select the **Add to album|Element info (Summary)** command.

 *Album button*

3. Select **Album** from the **View** menu – or click on the **Album** toolbar button – or press **Ctrl + A**. The new table should appear on a new Album page.

Note: Summary tables differ depending on the type of element. For example, we see an overflow rate for the settler summary and an OUR for a bioreactor. For more detailed instructions review the section on **Album Element Information Displays** in the "*BioWin Album*" chapter.

TUTORIAL 2B - A Nutrient Removal Refresher

This tutorial demonstrates application of BioWin to the system set up in Tutorial 2A. Aspects covered in this tutorial include phosphorus removal in an anaerobic selector, high-rate P removal systems, and high nitrification rate/high temperature conditions.

Anaerobic selector modification (including P removal)


File Open button

1. If you have restarted BioWin open the file **Tutorial 2** using the **File|Open** command – or click on the **Open** button on the toolbar.
2. We'll record results in the table below:

Temp	Nit. Rate	NML Recycle	SRT	Effluent Ammonia	Effluent Nitrate	Effluent PO4-P

3. Run the steady state simulation. Tabulate the results, and note the nitrification, denitrification and P removal performance.
4. You also need to record the SRT. Click on the **Project|Active SRT** command – or click on the **Active SRT** button on the toolbar. You can give this SRT a name if you like (to distinguish it in case you want to look at other SRT "scenarios", *e.g.* a case where the sludge mass in the settler is included in the SRT calculation). From the **Select elements for total mass** button, add all of the bioreactors. Click the **Select wastage elements** button, expand the **Effluent** tree, and select the **Wastage** element. The SRT will now appear on the status bar at the bottom of the screen.
5. Discuss possible retrofit options to achieve biological P removal in the existing tankage.
6. Try reducing the nitrified mixed liquor recycle rate (to zero).

Note: In specifying the wastage element(s) for the SRT calculation we selected the **Wastage output** element. We could also have selected the **side stream** (S) of the splitter where the waste stream is withdrawn. However, don't select both elements as this would count the wastage twice!

Note: We are calculating SRT based on the mass of sludge in the bioreactors. We could include the sludge in the clarifier.

Note: If we include the secondary effluent in the SRT calculation we would be accounting for solids lost via that stream.

High Rate P Removal System

1. Now we wish to modify the system to attain P removal without nitrification (with the mixed liquor recycle set to zero). To do this we increase the wastage rate to reduce SRT and wash out the nitrifiers. BioWin provides a convenient way to set the SRT to a specific value, and calculate the required wastage rate.

SRT Calculation/Control

2. Click on the **Project|Active SRT** command – or click on the SRT button on the toolbar. Place a check in the **Control SRT** box – new options appear in the lower part of the dialog.
3. Select **WAS splitter** from the pull-down list, and specify the last SRT from your table. Re-run the steady state simulation, and check that the wastage rate is 1 MGD.
4. Re-run for an SRT of 5 days, and tabulate the results. Reduce the SRT further to 4 days. Have we washed out the nitrifiers?

High Nitrification Rate / High Temperature Conditions

Now we encounter an unusually high temperature summer. What if the nitrification rate is high?

1. Change the maximum nitrifier growth rate from the value of 0.8 to 1.0/d.
2. Change the temperature to 24°C.
3. Repeat the simulation for an SRT of 4 days, and see if the nitrifiers still wash out.
4. Decrease the SRT further to 3 days, and repeat. Is the P removal performance good?

TUTORIAL 3 - Nitrification Dynamics and Setting up Charts

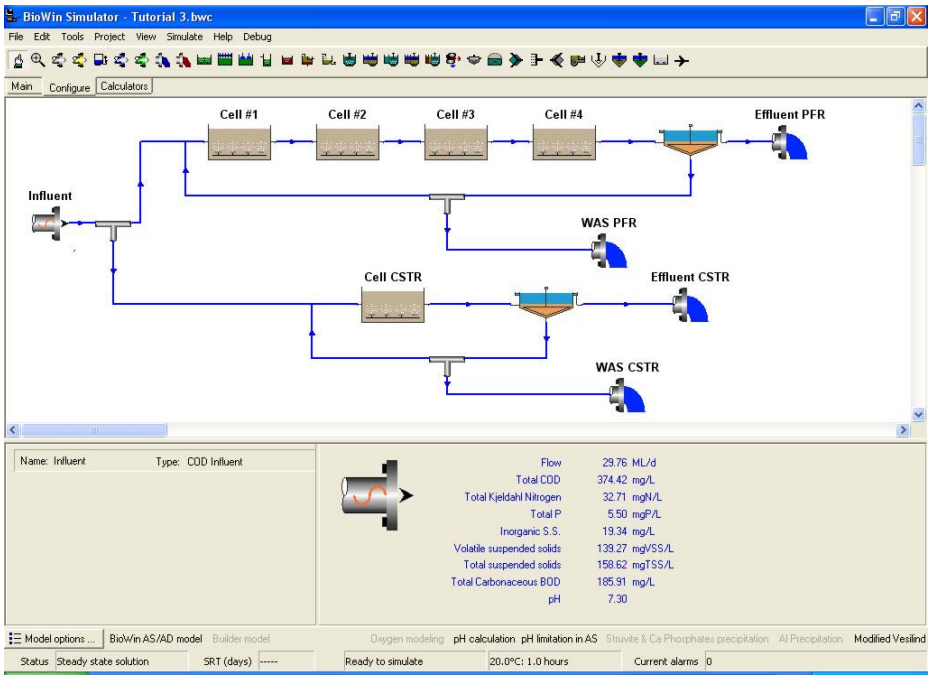
This tutorial demonstrates dynamic simulations and a comparison of nitrification performance in a plug flow versus a completely mixed reactor configuration. We will learn how to set up charts in the Album.

Aspects covered in the tutorial include steady state and dynamic simulations, and setting up charts in the Album.

The tutorial 3 system and the influent data

For this demonstration we will split the influent flow (actual field data) equally between two parallel trains as shown in the BioWin screen view below. The system has the following characteristics:

PFR reactors:	Four (each 1.2 ML)	Depth = 4.5 m	DO = 2 mg/L
CSTR reactor:	One (4.8 ML)	Depth = 4.5 m	DO = 2 mg/L
Clarifier (Ideal):	Each	Area = 1,000 m^2	Depth = 4.8 m
Influent:	Accept default wastewater characteristics		
RAS recycle:	Each	7.5 ML/d (50%)	
Wastage rate:	Each	0.2 ML/d (constant rate)	
Temperature:	20°C		
Nitrification rate:	0.9 /d		



The Tutorial 3 system configuration layout

The following diurnal influent loading pattern for flow, COD, TKN, TP and ISS has been established.

Time	Flow (ML/d)	COD (mg/L)	TKN (mg/L)	TP (mg/L)	ISS (mg/L)
0	29.8	437	28.7	5.0	17
2	20.4	401	29.2	5.3	12
4	14.4	333	29.7	5.1	13
6	14.3	341	29.8	4.8	8
8	23.9	260	24.1	3.7	9
10	37.6	279	33.5	4.7	16
12	41.9	402	42.9	7.0	25
14	40.5	383	40.5	6.9	27
16	35.0	419	36.2	6.5	25
18	32.6	411	31.8	5.9	18
20	32.7	364	27.8	4.8	22
22	34.0	406	26.2	4.4	21

Set up the configuration and influent data

1. Select a unit basis of ML and ML/d.
2. Create the configuration shown above, and set up all of the physical and operational data.
3. Double click on the influent element, click on the **Edit data** button, and enter the time-varying influent data recorded in the table above.

Hint: To save on typing all those numbers, open the Word document containing the table (**Tutorials.doc**), copy columns of data to the clipboard, and paste the data into BioWin. When pasting a column of data, place the cursor in the top cell in a column.

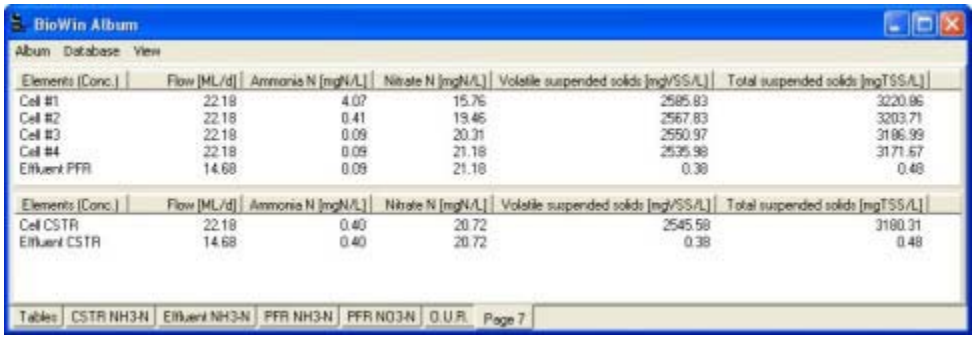
4. Save the file as **My Tutorial 3** in the **Data/Tutorials** directory.
5. Note the max. spec. growth rate from the **Project|Parameters|Kinetic** menu.

Note: In this system we do not include mixers for the influent and RAS streams. Both streams are connected directly to the front-end reactors in each train.

Steady State Performance

1. Run the steady state simulation.
2. Open the Album and add a new page with two horizontal panes using the **Album|Add page** command.

3. In the upper pane, set up a table that shows flowrate, NH3-N, NO3-N, VSS and TSS for the upper section (plug flow part) of the plant; that is, in each reactor and in the effluent.
4. Create a similar table in the lower pane, but for the lower section (completely mixed).
5. Re-run the steady state simulation, and discuss the results.
6. The tables on Page 7 (note that your page number may be different if you've started with a blank album) should be similar to that shown below.



Your results after running a steady state

Setting up charts

There are many options for creating different types of chart in BioWin. This tutorial will only show a few examples and a limited number of formatting options. These will be the ammonia and oxygen utilization rate responses in the system. More example charts are included in the **An Example** file. For details on charting options refer to the chapters:

- "*Creating Charts & Adding Series*"
- "*Chart Formatting Procedures*"
- "*Series Formatting Procedures*"

Note: The default chart template can be customized.

Create a Time series chart

In this section we set up a time series chart showing the ammonia concentration response in the completely mixed reactor, Cell CSTR, and in the effluent from this train.

1. Open the Album and use the **Album|Add page** command to add a page
2. Right click on the blank pane, and select the **Chart** command.
3. Position the cursor over the chart, right click, and select **Add Series**. This opens on the **Time series** tab – the one we want for now.
4. Select **Cell CSTR** in the **Element name** pull-down list, highlight **NH3-N** from the **State variables** list, and click the **Plot selected**

button. Select the **Line** option from the **Time series gallery**, and press **OK**.

5. Repeat this procedure to plot the effluent ammonia response. Select **Effluent CSTR** in the **Element name** pull-down list, highlight **NH3-N** from the **State variables** list, and click the **Plot selected** button. Select the **Line** option from the **Time series gallery**, and press **OK**.
6. Click the **Close** button in the **Add Series** dialog box.
7. On Page 2 (once again, your page numbering may be different – this is not a concern) of the Album set up a time series chart comparing the effluent ammonia concentrations for each of the two trains.

Note: No lines appear in the chart yet. You must first run a dynamic simulation (see below).

Note: Setting up charts automatically adds the plotted items to the database.

Create a Multi time series chart

In this section we set up a multi time series chart showing the ammonia concentration response in each of the reactors in the plug flow train, Cells #1 to #4. The multi time series chart is a variation of the time series where you plot the same variable in multiple zones simultaneously.

1. Open the Album and use the **Album|Add page** command to add a page
2. Right click on the blank pane, and select the **Chart** command.
3. Position the cursor over the chart, right click, and select **Add Series**. Select the **Multi time series** tab.
4. Expand the **Elements** tree in the window on the left, and move the four bioreactor cells to the right **Selected elements** list by selecting each and clicking on the right-pointing arrow. Select **NH3-N** from the **Compound** pull-down list, and click the **Plot selected** button. Select the **Line** option from the **Time series gallery**, and press **OK**.
5. Click the **Close** button in the **Add Series** dialog box.

Current value chart

1. In this section we set up a current value chart showing the nitrate concentration in each of the reactors in the plug flow train, Cells #1 to #4. A current value chart can be presented as an area, bar or pie plot.
2. Open the Album and use the **Album|Add page** command to add a page
3. Right click on the blank pane, and select the **Chart** command.
4. Position the cursor over the chart, right click, and select **Add Series**. Select the **Current value** tab.
5. Expand the tree in the **Elements** window on the left, and move the four bioreactor cells to the right **Selected elements** list. Select **NO3-N** from the pull-down **Compound** list, and click the **Plot**

selected button. Select the **Bar** option from the **Current value series gallery**, and press **OK**.

6. Click the **Close** button in the **Add Series** dialog box.
7. Right click on the legend and select the **Edit Legend** command. Change the **Text** style to **Point name only**.

Your own time series charts

Try setting up two time series charts (line plots) on one page in the Album.

1. Open the Album and use the **Album|Add page** command to add a page. Select the layout with two horizontal panes.
2. In the upper pane set up a time series chart for the OUR (total) response in the four-in-series reactor plug flow train.
3. In the lower pane set up a time series chart for the OUR (total) response in the single reactor completely mixed train.

Dynamic simulations

Dynamic Simulation button

From the main simulator window, set the dynamic simulation running for 1 day either from the **Simulate|Dynamic** simulation menu command or by clicking on the **Dynamic simulation** toolbar button. After pressing the start button select the **Start from** and **Current values** options, and a simulation time of **1 day**.

1. When the dynamic simulation is complete press the stop button in the player dialog.

Note: You can switch to the Album while the simulation runs.

2. View the simulation results in the Album.

Note: All of your time series plots are 3-D at this point – great for presentations! However, it may be simpler to interpret results from 2-D plots. On the last page of the Album right click on the charts, select the **Edit Options** command and on the **3 D** sub-tab of the **Chart** tab, uncheck the **3 Dimensions** box.

3. Continue the simulation for another 4 days. Set the dynamic simulation running from the **Simulate/Dynamic simulation** menu command or by clicking on the **Dynamic simulation** toolbar button. After pressing the start button select the **Continue from** and **Current values** options, and set the simulation time to 4 days.
4. Open the Album while the simulation runs.
5. When the dynamic simulation is complete press the stop button in the player dialog.
6. View the results in the Album and discuss options for reducing "break-through" of ammonia.
7. Re-run the steady state simulation and then run the dynamic simulation for 2 days using the **Start from** option. When the simulation is paused at 2 days, double click on each wastage splitter in the drawing board

and reduce each wastage rate from 0.2 to 0.1 ML/d. Press the start button and continue the dynamic simulation for 8 days.

Hint: Try starting a dynamic simulation by pressing the **F7** key when the **Album** is open.

Editing the charts

Move the dialog box to one side of the chart. That way you will see the changes happen immediately without having to close the dialog.

In setting up the charts you accepted many default charting options. For example, automatic scaling of axes, the increments on axes, the grid appearance, the color selection, the legend format, the chart titles, etc. The options are too numerous to list.

Experiment with the many chart options. Start by right clicking on a chart and selecting the various commands.

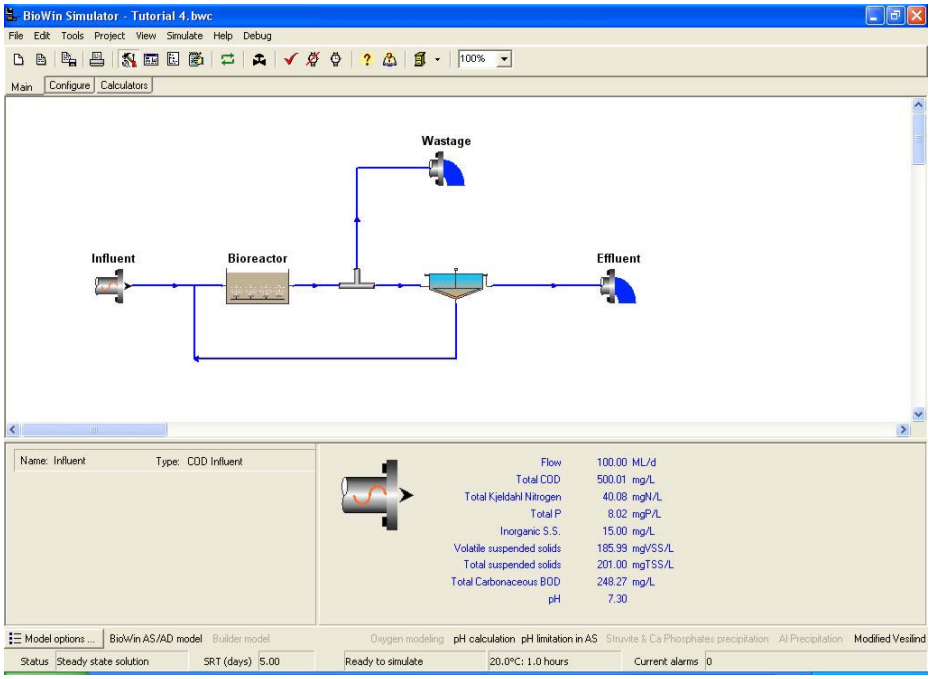
TUTORIAL 4 – Secondary Clarifier Simulation

This tutorial demonstrates aspects of modeling secondary clarifier performance with the one-dimensional settler model. Aspects covered in this tutorial include model settler behavior under steady state and dynamic conditions.

The tutorial 4 system

For the demonstration we set up a simple one-reactor system with a model settler as shown in the BioWin screen view below. The system has the following characteristics:

Bioreactor:	30 ML	Depth = 4.5 m	DO = 2 mg/L
Clarifier (Model):	Area = 4,000 m^2	Depth = 4.0 m	
RAS recycle:	Initially 100 ML/d (100%)		
Wastage rate:	6 ML/d (constant rate)		



The Tutorial 4 system configuration

1. Change to SI units (ML and ML/d) and set up the system.
2. Double click on the influent element, click on the **From file** button, and load the file **An Example.ifd** from the **DATA** directory.
3. Run a steady state simulation to check that you have specified all the necessary data.
4. Use the **File|Save As** command to save the configuration as **My Tutorial 4** in the **Data/Tutorials** directory.

Note: In this example the wastage stream effectively is withdrawn from the bioreactor, not the underflow. This is termed hydraulic SRT control. The reason for choosing this mode is that, irrespective of the underflow rate and the underflow TSS, the reactor TSS concentration will remain relatively constant. By wasting mixed liquor from the reactor we will maintain a relatively constant SRT even when the underflow rate changes. In this case wasting 6 ML/d from a bioreactor volume of 30 ML translates into a 5 day SRT (but remember we aren't accounting for sludge in the settler in this SRT calculation).

Recording results and modifying the Album

1. Add a page to the Album with two vertical panes. Display the element information (summary) for the bioreactor and the model settler in the two panes. Note: this step perhaps is not necessary – you'll be able to get all the required information from the main simulator window. (TSS values, settler solids loading rate – SLR, settler specific overflow rate – SOR, etc.)
2. We'll record simulation results in the table below. All of this information can be found either in the two-pane page you added to the

Album or by moving the cursor over elements in the drawing board and noting values displayed in the lower right pane.

Underflow Rate	Max Compactability	SLR	SOR	Effluent TSS	Reactor TSS	Underflow TSS

Setting up a settler profile in the Album

1. We want a view of the TSS concentration profile in the settler. Move the cursor over the settler in the drawing board, right click, and select the **Add to Album** command. Select **Profile Plot…** from the flyout menu. In the dialog highlight Total suspended solids in the right hand **Combined** list, select **Current values** for the profile type, and the **Concentration on X** orientation option. Click on the **Plot selected** button, select **Line** in the **General series gallery**, and close the dialog.
2. The preceding step generated a new plot in the Album. Open the Album – the new page should be visible – and confusing! In the plot, concentration is on the x-axis (as selected above), and settler height is on the y-axis *with the overflow concentration at the bottom – hence the confusion*. We should invert the y-axis to make the plot more sensible.
3. Right click on the chart and select the **Edit Axes** command. For the left axis, place a check in the **Inverted** box on the **Scales** sub-tab. Press the **Close** button.

Note: In this case we are simulating the settler as 10 layers in the vertical direction – numbered from top to bottom as 0 to 9.

4. While we are editing the chart we should change the bottom axis (concentration) scale to have a minimum and maximum of 15,000 mg/L, respectively. Right click on the chart and select the **Edit Axes** command. For the *bottom* axis, in the **Scale** tab uncheck the **Automatic** box, and use the **Change…** buttons to specify the Maximum and Minimum axis values. Press the **Close** button.

Steady state simulations

1. Run the steady state simulation. Note the effluent and underflow TSS, and the SLR and SOR. View the settler profile in the Album.
2. Change the underflow rate to 50 ML/d by double clicking on the settler in the drawing board and going to the **Flow split** tab. Repeat the steady state simulation and record the results.

3. Change the underflow rate to 33 ML/d by double clicking on the settler in the drawing board and going to the **Flow split** tab. Repeat the steady state simulation and record the results.

Dynamic Simulations

1. Add a page to the Album with two horizontal panes. In the upper pane set up a time series chart (**Line** style) for the settler specific overflow rate, SOR. Set minimum and maximum values on the left axis of 0 and 20. In the lower pane set up a time series chart (**Line** style) for the settler solids loading rate, SLR. Set minimum and maximum values on the left axis of 0 and 200.

Note: Initially the charts will be blank because we have yet to run a dynamic simulation.

2. Add another page to the Album with two horizontal panes. In the upper pane set up a time series chart (**Line** style) for the effluent TSS. Set minimum and maximum values on the left axis of 0 and 30. In the lower pane set up a time series chart (**Line** style) for the settler underflow TSS. Set minimum and maximum values on the left axis of 0 and 16,000.
3. Start a dynamic simulation for 2 days. Observe the predicted performance of the model settler in the Album.

Hint: If you start the simulation from the BioWin main window the Album disappears. You can keep it open while the simulation is running if you use the F7 key to start the simulation.

4. When the simulation is paused change the RAS rate to 100 ML/d. Continue the simulation for another 3 days.
5. When the simulation is paused change the RAS rate to flow-paced at 33% (based on the influent flow). Continue the simulation for another 3 days.
6. From the **Project|Parameters|Settling...** menu, on the **Modified Vesilind** tab, change the maximum sludge compactability to 8,000 mg/L. Continue the dynamic simulation for another 6 days. Watch the settler profile and effluent TSS in the Album.
7. Try other situations with changes to the settler area/depth, and changes to the sludge settling properties.

Note: Setting a low sludge compactability may cause problems with steady state simulations not converging. This is a result of numerical solver problems because there can be multiple solutions to the mass balance equations. In this situation, what you may wish to do in place of a steady state simulation is run a dynamic simulation for an extended period of 3 - 4 SRTs. This should move to the steady state solution.

TUTORIAL 5 - Aeration System Simulation

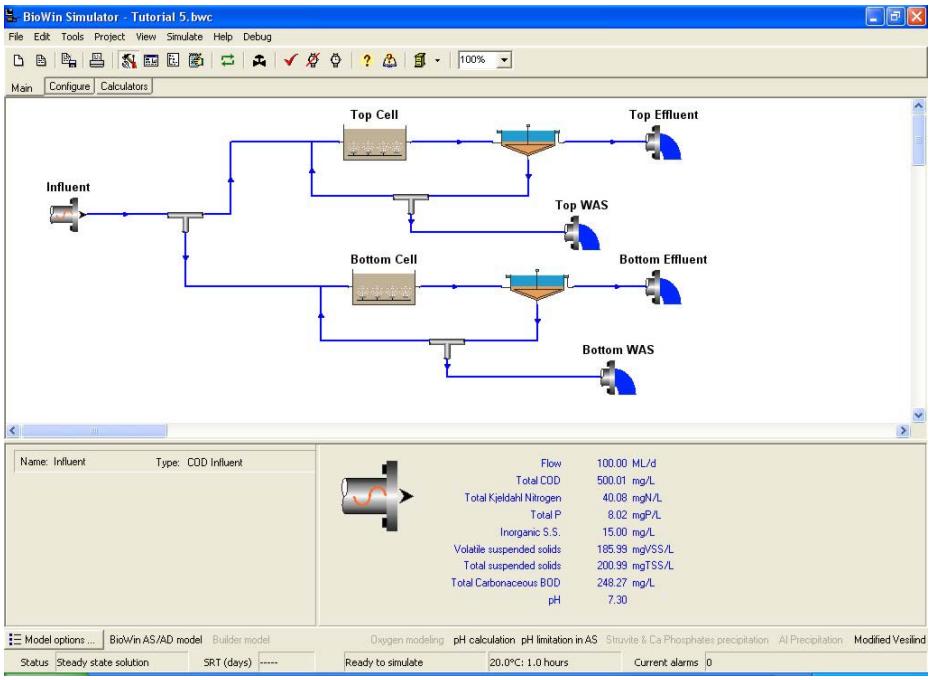
This tutorial demonstrates diffused aeration system modeling.

Aspects covered in this tutorial include effects of changing oxygen parameters under steady state and dynamic conditions.

The tutorial 5 system

For the demonstration we will split the influent flow equally between two parallel trains as shown in the BioWin screen view below. The systems have the following characteristics:

Bioreactors:	Each	25 ML	Depth = 3.0 m	DO = 2 mg/L
Clarifier (Ideal):	Each	Area = 2,000 m^2	Depth = 4.0 m	
RAS recycle:	Each	50 ML/d (100%)		
Wastage rate:	Each	1.0 ML/d (constant rate)		



The system used for Tutorial 5

1. Change to SI units (ML and ML/d) and set up the system.
2. In the influent element, load the file **An Example.ifd** from the **DATA** directory
3. Use the **File|Save As** command to save the configuration as **My Tutorial 5** in the **Data/Tutorials** directory.

Recording results and modifying the Album

1. Add a new page to the Album with two vertical panes. Display the element information (summary) for the top reactor and the bottom reactor in the two panes.

2. We'll record simulation results in the table below. All of this information can be found in the two-pane page you added to the Album.

Depth	Temp.	αF	DO	OUR	Q_{AIR}	OTR	SOTE(%)
3.0	20	0.5	2				
4.5	20	0.5	2				
6.0	20	0.5	2				
4.5	12	0.5	2				
4.5	20	0.5	4				
4.5	20	0.8	2				

Steady state simulations

1. Run a steady state simulation and evaluate the predicted performance of the aeration system. [Note: data for the top and bottom reactors should be the same. If not, you must have an error because the two systems should be set up identically, each receiving half of the influent flow]. Tabulate one set of the results.
2. Change the depth of the lower bioreactor in steps from 3.0 m to 4.5 and then 6.0 m. For each change, re-run the steady state simulation, and tabulate the results for the new depth.
3. Set the depth of each bioreactor to 4.5 m. The global temperature for the system is 20°C. Double click on the bottom cell to open the **Properties** dialog. On the **Operation** tab, check the local temperature option, and specify a temperature of 12°C. Re-run the steady state simulation, and tabulate the results for the new temperature.
4. Re-set the temperature to 20°C. Change the DO setpoint in the lower bioreactor to 4 mg/L, re-run the steady state simulation, and tabulate the results.
5. Re-set the DO setpoints to 2 mg/L in each reactor. Change α for the lower reactor to 0.8 (double-click on the **Bottom Cell**, click the **Operation** tab, check the **Local aeration parameters box** and click the **Model Parameters** button – you can find aeration parameters such as alpha and beta on the **Aeration** tab). Re-run, and tabulate the results.
6. Switch on oxygen transfer and DO modeling in the **Project|Current Project Options** menu on the **Model** tab. Re-run the steady state simulation, and discuss differences in model predictions.
7. Re-set the α values to 0.5. Instead of specifying DO setpoints, switch to air supply rate and adjust the values.

Dynamic Simulations

1. Modify the Album to include charts of air supply rate, DO and OURt for each of the aerated reactors.
2. Start a dynamic simulation. Observe the response of the aeration parameters.
3. Pause the simulation and change aeration parameters. Attempt to predict the effect of the change before continuing the simulation.

TUTORIAL 6 – Setting Up an SBR System

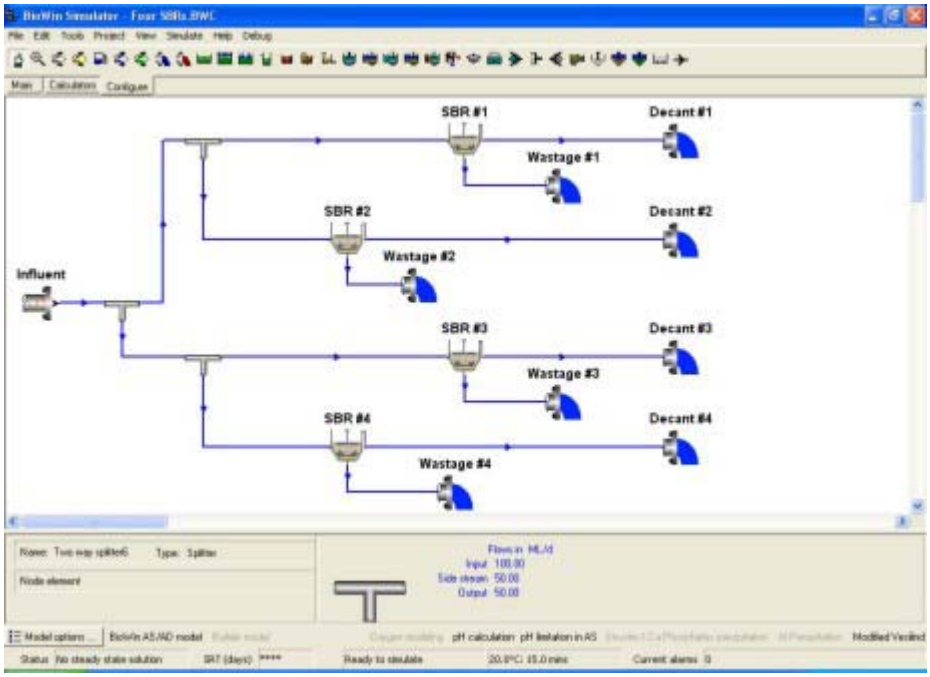
This tutorial provides an introduction to using sequencing batch reactor (SBR) modules in BioWin. The system considered here is based on the simplest SBR module; namely, a single-tank unit (without prezones) where filling, settling and decanting all occurs in one zone without baffles.

Important Notes:

- This configuration is not proposed as a potential design. Rather, the objective merely is to illustrate how to set up an SBR system in BioWin.
- Detailed Help on the different SBR modules is provided in the *Element Types*"chapter.
- There are certain basic issues to consider when simulating SBR systems. See the introductory notes of the **More Examples** section later in this document.

The System

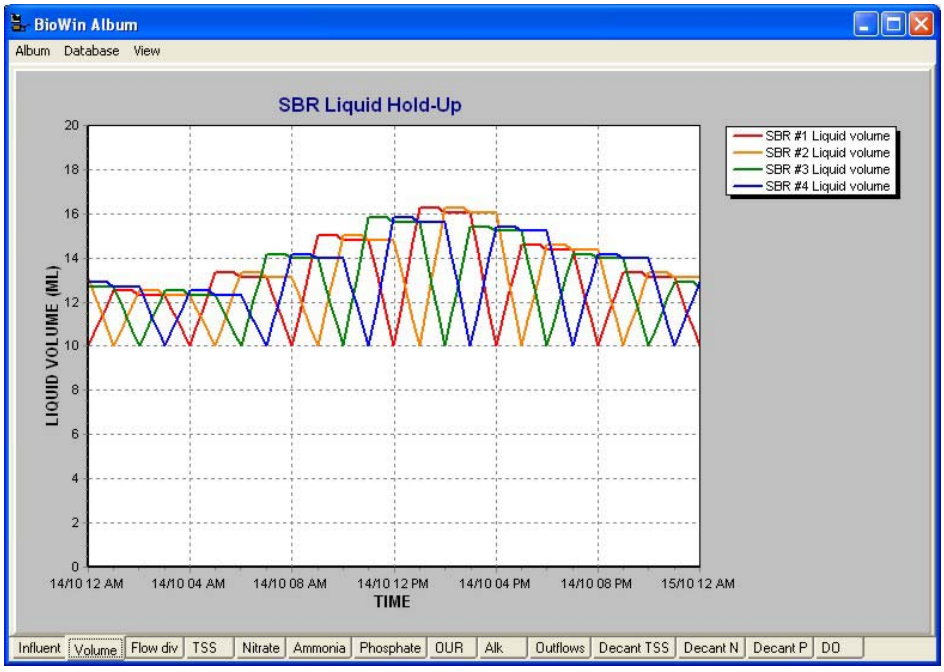
We wish to set up an SBR system with four identical units in parallel. The layout of the system is shown in the figure below. Each unit will operate on the same cycle, with equal periods for fill, react, settle and decant (1 hour each, with a total cycle time of 4 hours, and 6 cycles per day). Influent to the system is directed sequentially to each of the four units for a fill period of one hour. That is, SBR #1 receives influent for 1 hour (while SBR #2 is decanting, SBR #3 is settling, and SBR #4 is reacting). At the end of the hour influent is directed to SBR #2 and each of the SBRs moves to the next stage in its cycle (SBR #1 starts the react period, SBR #3 starts decanting, and SBR #4 starts settling). In summary, each SBR is operated on the same cycle, but the cycles are offset from the adjacent unit by one hour.



The four-unit SBR system configuration

Suggested Approach

The response and interactions in a four-unit system like this can be very complex. Even viewing the results can be confusing! As an example, the chart below shows how the level in each SBR might change over 24 hours. It is strongly suggested that the practical approach is to first set up a system that includes the flow division, but with only one of the trains; that is, only one SBR unit. The system then can be debugged more readily.

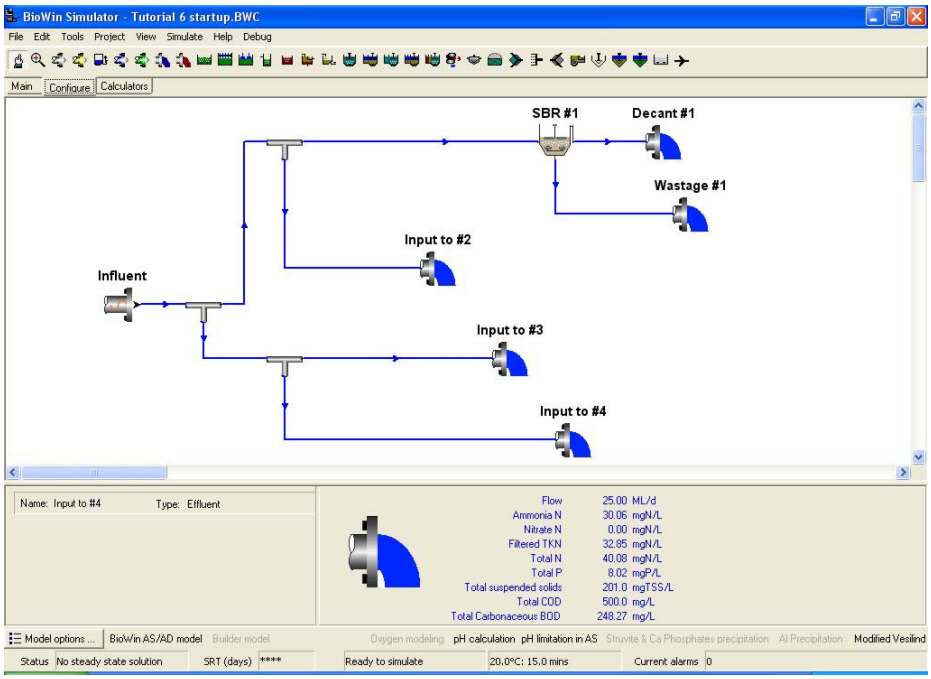


Volume plot for system incorporating all 4 SBRs

Setting up the One-Unit Configuration

Set up the single-tank SBR system shown in the BioWin screen view below. The characteristics that we will specify for the system are as follows:

SBR (full) volume	20 ML
Depth	4.5 m
Width	66 m
Minimum decant level	50%
Cycle:	Fill 1 hour
	React 1 hour
	Settle 1 hour
	Decant 1 hour
DO (fill and react phases)	2 mg/L
Initial liquid hold-up	50%
SBR underflow (for wastage):	20 ML/d from 1:45 to 2:00 hours in each cycle.



The Tutorial 6 system configuration

1. Change to SI units (ML and ML/d) and set up the system.
2. Right click on elements to change names.
3. Double click on the influent element, click on the **From file** button, and load the file **An Example.ifd** from the **DATA** directory.
4. Use the **File|Save As** command to save the configuration as **My Tutorial 6** in the **Data/Tutorials** directory.

Specifying the flow distribution information

1. Double click on the first flow splitter element after the influent – or click the right mouse button and select the **Properties** command.
2. Select the **Flow split** tab, and specify the splitter as a **flow router** by placing a check in the box at the lower left. Click on the **Routing pattern** button, and specify that the flow is **Switched at intervals** of 2 hours. This directs all the influent to SBRs #1 and #2 for 2 hours, then to SBRs #3 and #4 for 2 hours, and so on.
3. For each of the flow splitters in front of a pair of SBRs, double click on the element – or click the right mouse button and select the **Properties** command. Select the **Flow split** tab, and specify the splitter as a **flow router** by placing a check in the box at the lower left. Click on the **Routing pattern** button, and specify that the flow is **Switched at intervals** of 1 hour. Any flow reaching the router from the upstream router is alternated between the two SBRs at one-hour intervals.

File Save Button

4. When you are finished use the **File|Save As…** command – or click on the Save button on the toolbar - to save the configuration as **My Tutorial 6** in the **Data/Tutorials** directory.

Specifying the SBR physical information

1. Double click on the SBR element – or click the right mouse button and select the **Properties** command.
2. On the **SBR dimensions** tab enter the SBR volume (full), depth and width (20 ML, 4.5 m, 66 m).
3. On the **Initial values** tab enter the initial SBR liquid hold-up as 50% (of the maximum i.e. 10 ML).

Specifying the SBR operational information

For the SBR operation we wish to specify equal 1 hour periods for fill, react, settle and decant; that is, a 4 hour cycle. The fill and react phases define the "mixed" period of operation, where it is assumed that the reactor contents are well-mixed through either aeration or mechanical mixing.

Hint: It is simplest if you adjust the cycle times in the order **Mix start time**, **Decant start time**, **Cycle length**. BioWin is continually checking your input data, and won't allow things like decanting after the end of the cycle.

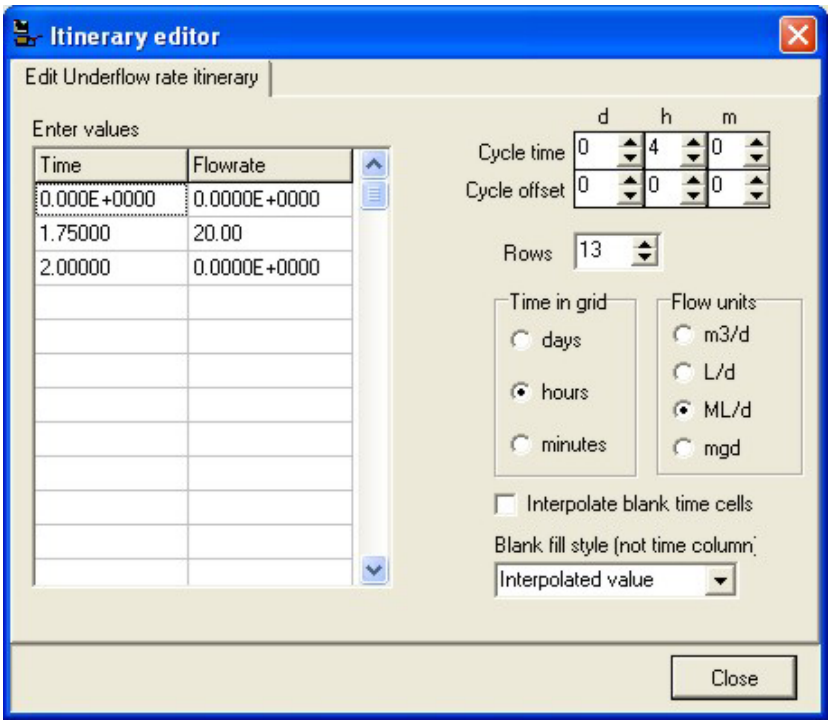
Hint: When reducing times from 1:00:00 (i.e. 1 day, zero hours, zero minutes) to say 4 hours, first increase the number of hours to 4 (i.e. 1:04:00) and then reduce the day unit (i.e. 0:04:00).

1. On the **SBR operation** tab start by setting the **Mix until / Start settling at** time to 2 hours (i.e. fill + react).
2. Set the **Decant / Draw starting at** time to 3 hours.
3. Set the **Cycle length or duration** to 4 hours.
4. Select the **To minimum decant level** option so that the SBR is decanted to 50% each cycle.
5. Click on the **SBR aeration** button and specify a constant **DO setpoint** of 2 mg/L.

Specifying the sludge wastage information

We wish to waste sludge from the SBR at a rate of 20 ML/d for a period from 1:45 to 2:00 during each cycle (i.e. 1.75 to 2.00 hours in decimal format). This corresponds to a waste volume of 208 m^3 per cycle.

1. On the **SBR underflow** tab select the **Flow pattern** option and click on the **Pattern** button.
2. Select **hours** in the **Time in grid** group.
3. Set the **Cycle time** to 4 hours.
4. In the grid enter the time and flow data as shown below.



The SBR underflow tab showing the wastage information

File Save Button

Note : All the required information has been specified. When you are finished use the **File|Save As…** command – or click on the Save button on the toolbar - to save the configuration as **My Tutorial 6** in the **Data/Tutorials** directory.

Checking the system set-up

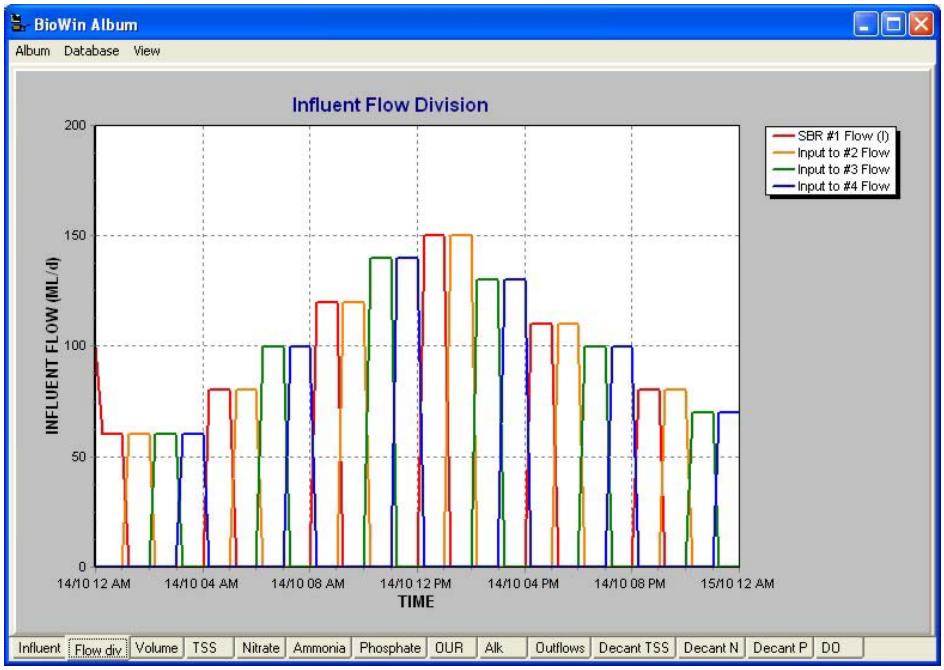
We want to check that all the data has been specified correctly. Generally this is achieved most easily by (1) following the SBR liquid hold-up over a 24 hour period (six cycles), and (2) checking that the flow routers are distributing the influent flow in the correct sequence. [In this example the influent flow rate varies over the 24 hour cycle, but is specified as constant for two-hour periods. This simplifies checking the SBR volume response].

Note: No lines appear in the charts until you run a dynamic simulation (see below).

1. Open the Album on **Page 1**.
2. Right click on the blank pane, and select the **Chart** command.
3. Position the cursor over the chart, right click, and select **Add Series**. This opens on the **Time series** tab – the one we want for now.
4. Select **SBR #1** in the **Element name** pull-down list, highlight **Liq. Vol.** from the **State variables** list, and click the **Plot selected** button. Select the **Fast line** option from the **Time series gallery**, uncheck the **3D** option, and press **OK**.
5. Click the **Close** button in the **Add Series** dialog box.
6. Repeat this procedure to plot the flow to each of the four parallel branches in Page 2. Do this by sequentially selecting the **Output** and the **Sidestream** of each flow router as the series to plot.
7. Repeat this procedure to plot the TSS concentration in SBR #1 in Page 3.
8. Repeat this procedure to plot the influent flow rate in Page 4.
9. Rename the Album pages by right clicking on the tabs at the bottom of the Album.
10. From the main simulator window, select the **Project|Database|Data interval** option, and change the interval to 15 minutes. This is the interval at which monitored data are added to the database and charts.

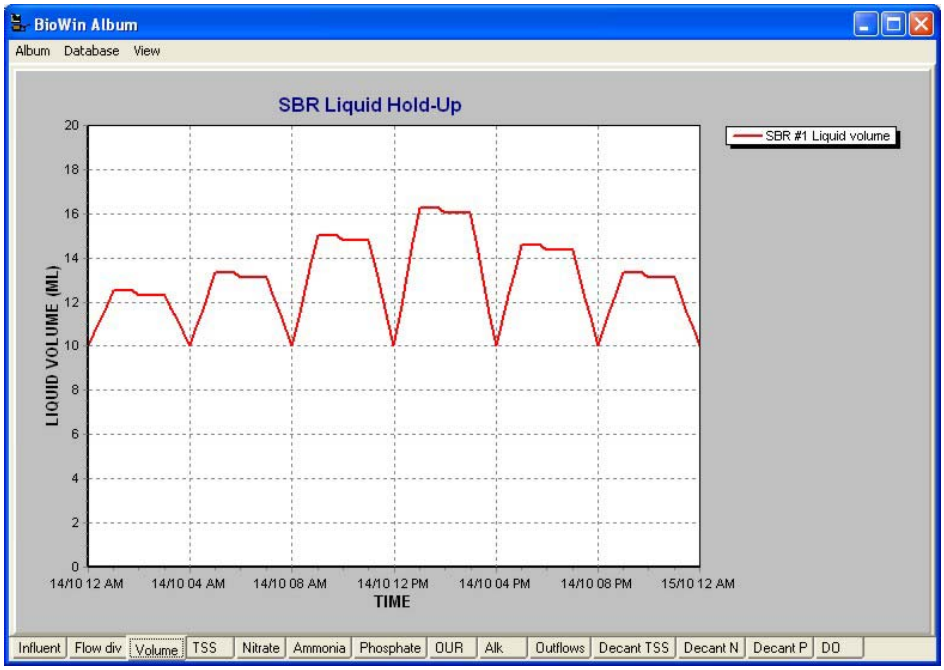
Dynamic Simulation Button

11. From the main simulator window, set the dynamic simulation running for 1 day either from the **Simulate|Dynamic simulation** menu command or by clicking on the Dynamic simulation toolbar button (or press the **F7** key). After pressing the start button select the **Start from** and **Seed values** options, and a simulation time of 1 day.
12. View the response in the Album. While the simulation is running open the Album by pressing **Ctrl + A**.
13. Check that each SBR receives influent flow for the appropriate one-hour intervals. The album chart should appear as shown below.



Influent flow distribution

14. Check that the SBR volume response is correct, as shown in the view below. There should be 6 cycles over the 24 hours. At the start of each cycle the volume should be 10 ML (50% hold-up). For the first hour the level increases. From 1:00 to 1:45 the level is constant, and then decreases by a small amount (208 m^3) when wasting occurs over the last 15 minutes of the mixing period. Decanting starts three hours into the cycle, and continues until the level reaches the 50% minimum at 4 hours (end of the cycle). During different cycles the extent of filling differs because the influent flow rate changes. It is worth checking that the volume increase during a fill period corresponds to the amount of influent over that one-hour period.

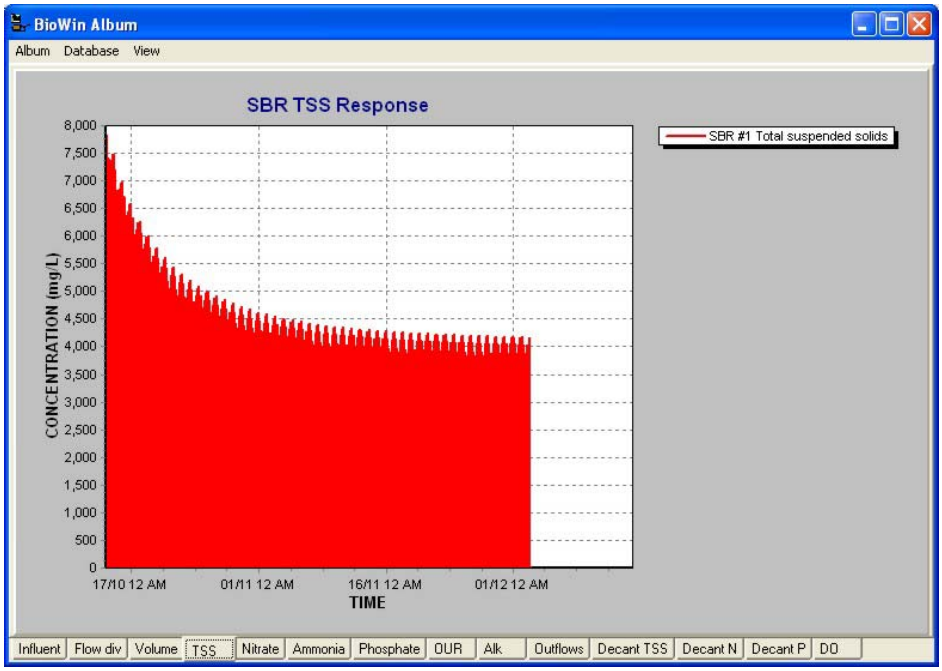


The SBR liquid volume response

Running the SBR simulation to reach a steady state

Dynamic Simulation Button

1. From the main simulator window, set the dynamic simulation running for 50 days either from the **Simulate|Dynamic simulation** menu command or by clicking on the Dynamic simulation toolbar button (or press the **F7** key). After pressing the start button select the **Start from** and **Seed values** options, and a simulation time of 50 days.
2. In the Album monitor the TSS response as the system approaches a steady state. While the simulation is running open the Album by pressing **Ctrl + A**. Right click on the chart and use the **Edit axes** option to select the **Automatic** option for the **Bottom** axis. This will automatically scale the chart.
3. When the dynamic simulation is complete press the stop button in the player dialog. For this system the response should have stabilized after approximately 40 days as shown in the view below. In this case the simulation **Start from** date was October 14.



TSS response over 50 days

Viewing the stable SBR response

Now that the system has reached a steady state typically we will want to view the response over 24 hours (or individual cycles). Usually it is most convenient to "set the clock back to time zero" (the **Start from** date of October 14 in this case), but start the simulation based on the **Current values**; that is the conditions after 50 days of simulation. This will save us from having to change the time axis repeatedly.

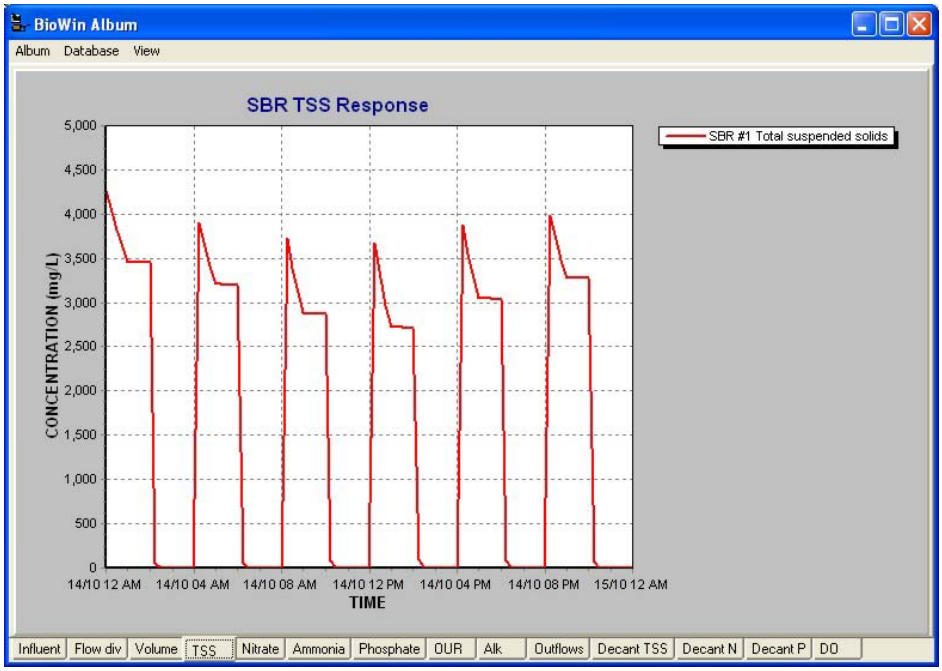
Dynamic Simulation Button

1. From the main simulator window, set the dynamic simulation running for 1 day either from the **Simulate|Dynamic simulation** menu command or by clicking on the Dynamic simulation toolbar button (or press the **F7** key). After pressing the start button select the **Start from** and **Current values** options, and a simulation time of 1 day.
2. When the dynamic simulation is complete press the stop button in the player dialog.
3. In the Album view the simulated SBR response. The view below shows the TSS response in SBR #1 over 24 hours (6 cycles). For the first hour in each cycle (during fill) the concentration decreases as the volume increases. From 2 to 3 hours (during react) the TSS remains near constant. When settling commences the plotted TSS decreases rapidly. This is the TSS concentration in the top layer of the SBR. The maximum TSS changes from cycle to cycle because the amount of fill (and dilution) changes.
4. Add additional charts to the Album.

File Save Button

5. At this point use the **File|Save As…** command – or click on the Save button on the toolbar - to save the configuration as **My Tutorial 6** in the **Data/Tutorials** directory.

Note: The file is being saved "as is". This means that at a later date the file can be loaded, and re-run using the **Start from** and **Current values** options (i.e. the status of the system at the time of saving the file). This obviates the need to run for an extended period to reach steady state



TSS response over 24 hours

The BioWin file for this system can be found in the **Data/Tutorials** directory under the name: **Tutorial 6.BWC**.

The Album includes charts for a large number of parameters.

Important note: Making changes to SBR physical or operational data often requires running the simulation for an extended period to attain a new steady state.

More BioWin Examples

A number of example configurations are provided on the installation CD. These can be found in the **Data\Examples** directory. These highlight some of the advanced features available in BioWin. This chapter provides a brief description of the systems. A level of familiarity with BioWin is assumed – detailed instructions for setting up the configurations such as those given in the Tutorials chapter are not given here. Therefore, it is recommended that you complete the tutorial exercises earlier in this chapter before investigating these examples.

Example SBR Systems

Notes :

- The example configurations are not proposed as potential designs. Rather, the objective is to illustrate how to set up SBR systems in BioWin, based on the different SBR modules.
- Detailed Help on the different SBR modules is provided in the "*Element Types*" chapter.
- There are certain basic issues to consider when simulating SBR systems. Read the **Considerations in SBR Systems** section before proceeding with the examples.
- All the examples use the influent loading pattern from the file **An Example.IFD**, and default wastewater characteristic fractions.
- The nitrifier maximum specific growth rate is set at 0.5 /day for all the examples.
- SRT for all of the systems is in the range of 15 days. Each case was simulated for 60 days to attain steady state, and then re-run for 24 hours using the **Start from** and **Current values** options before saving the files.

Important note: Making changes to SBR physical or operational data often requires running the simulation for an extended period to attain a new steady state.

List of SBR Examples

BioWin files for these systems can be found in the **Data/Examples/SBR** directory. The Albums include charts for a large number of parameters.

1. Four-in-Parallel Aerobic SBRs
2. Single Continuous Flow SBR with Unaerated Mixed Pre-zone
3. Single Continuous Flow SBR with Unaerated Settling Pre-zone
4. Single Continuous Flow SBR with Two Unaerated Mixed Pre-zones
5. Single Continuous Flow SBR with Two Unaerated Settling Pre-zones
6. Two-in-Parallel SBRs with Unaerated Mixed Pre-zone
7. Two-in-Parallel SBRs with Unaerated Settling Pre-zone
8. Two-in-Parallel SBRs with Two Unaerated Mixed Pre-zones
9. Two-in-Parallel SBRs with Two Unaerated Settling Pre-zones
10. Two-in-Parallel SBRs with Unaerated Constant-volume Pre-zone
11. Two-in-Parallel Single-Tank SBRs with Unaerated/Aerated Periods
12. Wastewater Characterization Fill-and-Draw SBR

Considerations in SBR Systems

Three aspects inherent to SBR systems require special consideration:

- Steady State
- Sludge Age
- Cycle Offsets

Steady state with SBR systems

SBR systems necessarily are dynamic, and the concept of steady state is not relevant. However, if we operate an SBR system with a fixed influent loading the SBR system will attain a quasi "steady state". [Note: the input could be either a Constant Input or a Variable Input]. That is, the SBR system response would be repeated identically from cycle to cycle, where the term "cycle" refers to the longest cycle in the dynamic system [not necessarily the cycle time of the SBR]. For example, the cycle time of a Variable Input influent element typically would be 24 hours, and this would be the quasi steady state cycle time (provided of course that the SBR cycle time is less than 24 hours which usually is the case). Even with a Constant Input, the quasi steady state "cycle" time may differ from the SBR cycle time. For example, perhaps wastage only occurs every third cycle; in this case the steady state "cycle" time will be three times the cycle time of the SBR.

A general guideline for the time a dynamic system takes to reach a quasi "steady state" is three to four times the longest time constant in the system. In the case of activated sludge systems usually the longest time constant will be the sludge age (or SRT). For SBR systems it is difficult to calculate an exact SRT so it is suggested that the appropriate procedure to identify when "steady state" is reached is by simulating for an extended period while viewing the TSS response.

When setting up an SBR system it is suggested that you first simulate for an extended period until you attain a quasi "steady state". Then save the file. When you re-open the file at a later time, you start off from the "steady state" [by running the dynamic simulation from **Current values** when the dynamic simulation is started.

Sludge age or SRT in SBR systems

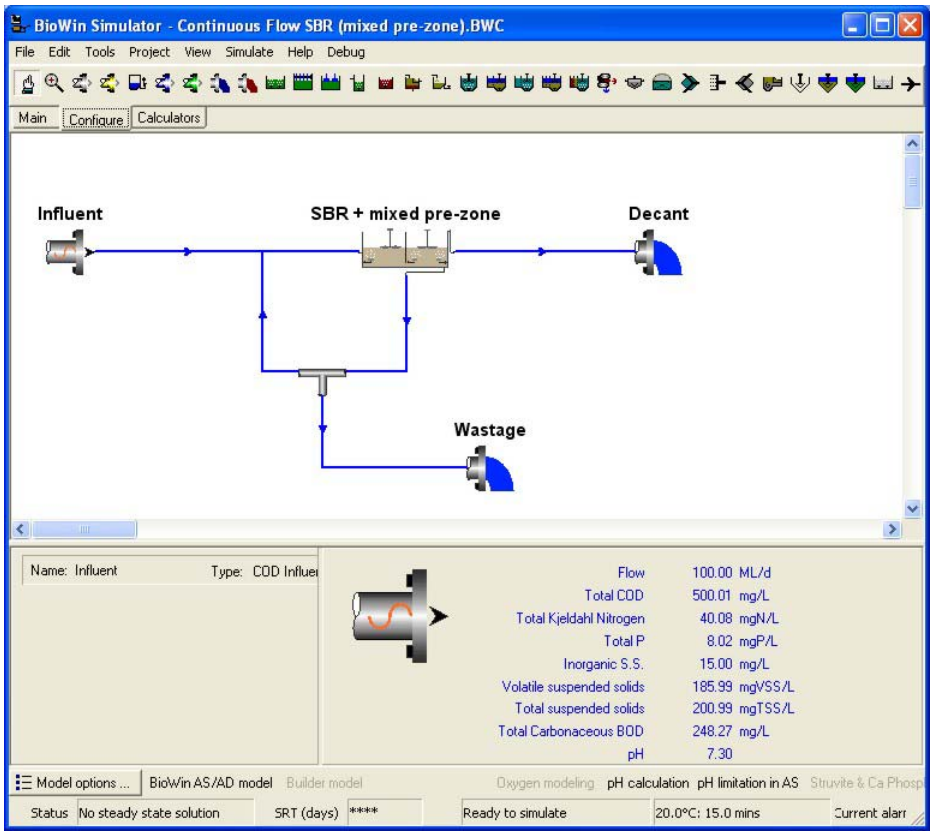
Sludge age (or SRT) is defined as the mass of sludge in the system divided by the mass of sludge wasted per day. In practice sludge likely will be wasted from an SBR towards the end of the settle/decant phase. At this time the sludge concentration is high and the waste volume will be lowest for a given mass wasted.

Design of activated sludge systems usually is based on some notion of a desired SRT. Experience with simulating SBRs indicates that it is difficult to estimate a required wastage rate for the target SRT if wasting occurs during settle/decant. This difficulty is related to a number of factors: quantifying the mass of sludge in the system, the sludge compactability, the changing sludge concentration at the base of the SBR during settle/decant, and the possibility of "rat holing" if the wastage rate is high.

For simulation purposes it is simpler to achieve a desired SRT with reasonable accuracy through setting up the system for hydraulic SRT control; that is, wasting of mixed liquor during the mixed phase rather than wasting settled sludge. For example, setting an SRT of 10 days requires wasting one tenth of the mixed reactor contents (volume) each day. Even this approach is not explicitly suited to SBR systems because the liquid hold-up and mixed liquor solids concentration likely changes from cycle to cycle. However, consideration of the liquid volume response usually allows a reasonable estimate of the volume of mixed liquor to waste so as to attain a given SRT. For example, consider an SBR with a maximum volume of 20

ML that is operated with 4 cycles per day, and decanted to 50% of the maximum each cycle. A trial simulation will provide an estimate of the average hold-up at the end of each cycle; say this is 16 ML. To attain an SRT of approximately 10 days requires wasting 1.6 ML of mixed liquor each day, or 0.4 ML/cycle. In this case we could set up the wastage to operate over the last 15 minutes of the mixed period in each cycle, at a rate of 1.6 ML/h (i.e. 38.4 ML/d for 15 minutes).

Hint: In certain SBR configurations there is recycling of sludge (usually only during the mixed phase) from the decant zone to an upstream pre-zone or reactor. This recycle can be set up using the SBR underflow. Wasting of mixed liquor is achieved via a splitter in the recycle stream, as shown in the view below.



SBR with recycle and wasting from the recycle

Cycle offsets

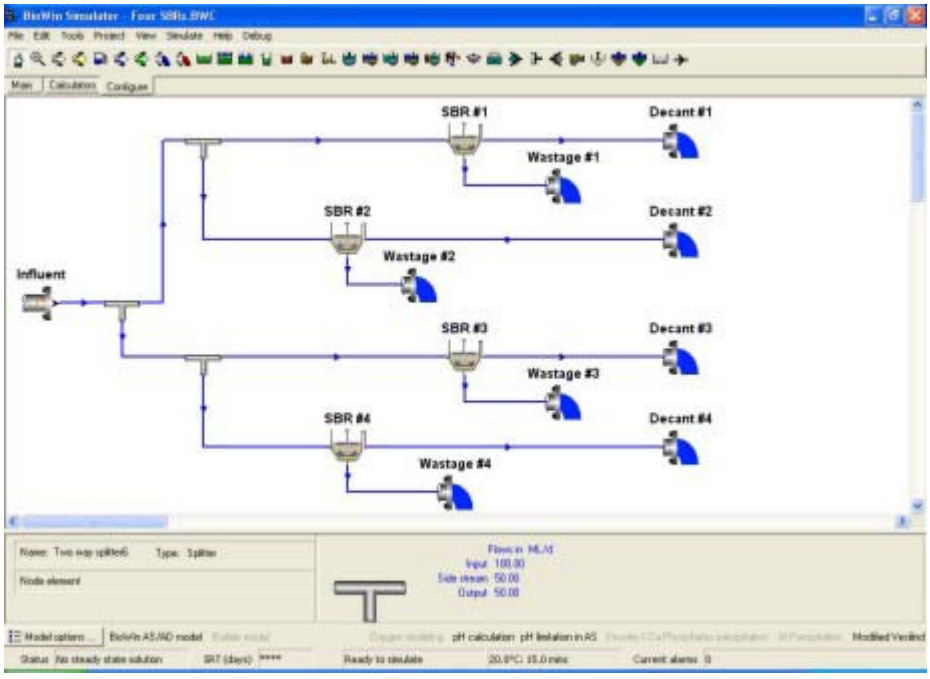
Individual units in parallel-unit SBR systems usually operate with the same cycle times of fill, react, settle and decant. However, the cycles are staggered in time. This is achieved readily in BioWin through offsets.

Hint: Select the unit that first receives influent as the reference (zero offset). An offset is specified for each of the other units. Think of the offset as the "time into the cycle" for that unit when the reference unit first receives influent.

Consider the four SBR system shown below. Each unit operates on the same cycle, with equal periods for fill, react, settle and decant (1 hour each, with a total cycle time of 4 hours, and 6 cycles per day). Influent to the system is directed sequentially to each of the four units for a fill period of one hour. That is, SBR #1 receives influent for 1 hour (while SBR #2 is decanting, SBR #3 is settling, and SBR #4 is

reacting). At the end of the hour influent is directed to SBR #2 and each of the SBRs moves to the next stage in its cycle (SBR #1 starts the react period, SBR #3 starts decanting, and SBR #4 starts settling). Offsets will be specified as follows: SBR #1 is the reference; SBR #2 must be 3 hours "into the cycle" because in one hour it will receive influent; and so on. In summary:

Unit	Offset (hours)
SBR #1	Zero
SBR #2	3
SBR #3	2
SBR #4	1



Four-in-Parallel System

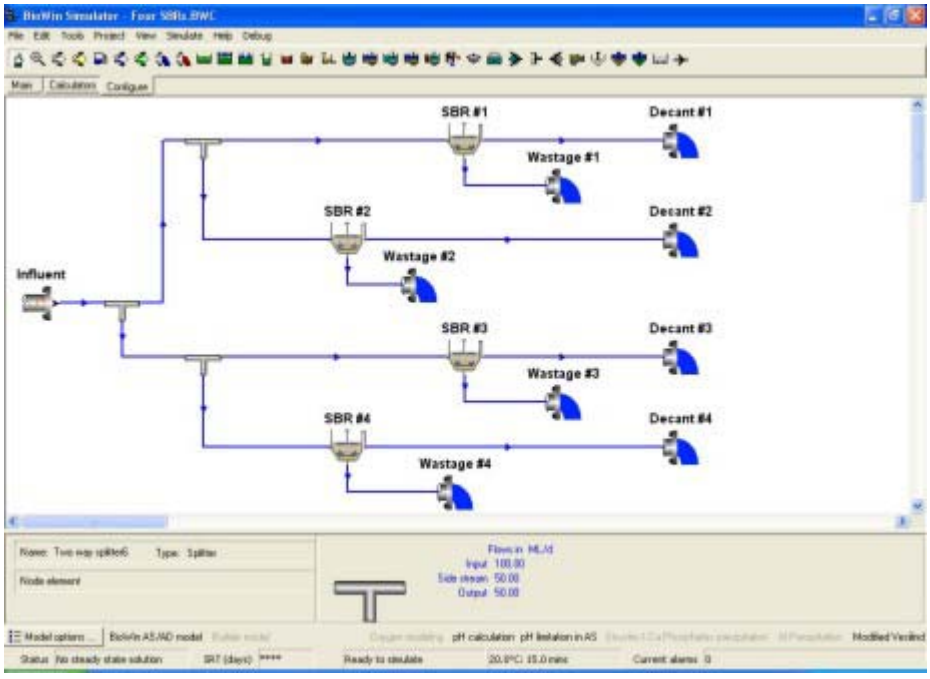
Four-in-Parallel Aerobic SBRs

The BioWin file for this system can be found in the **Data/Examples/SBR** directory under the name: **Four SBRs.BWC**.

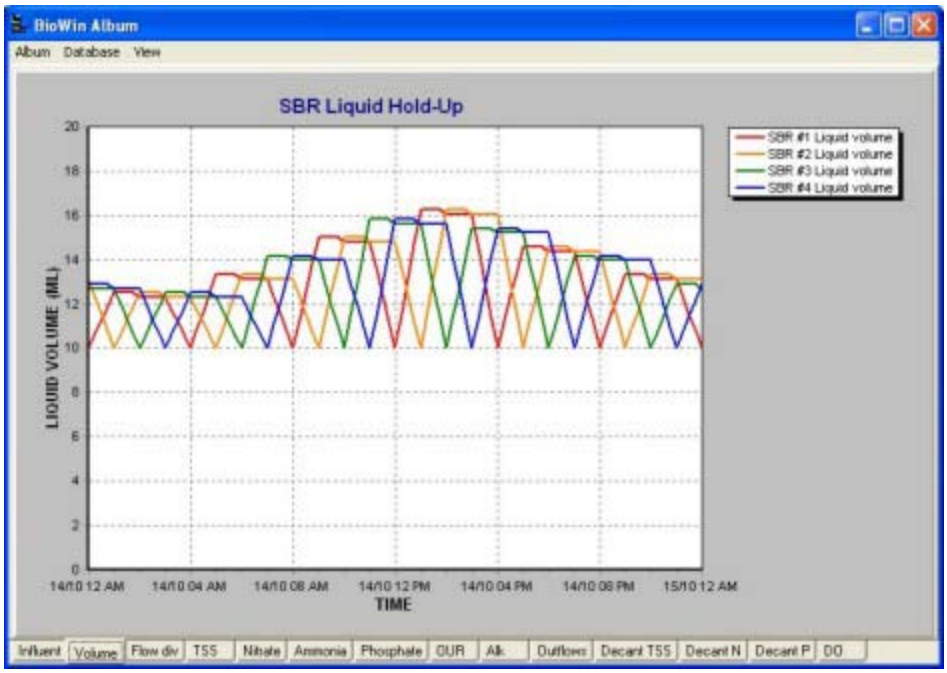
This example completes the system set up in **Tutorial 6**. There are four identical units in parallel. Each unit operates on the same cycle, with equal periods for fill, react, settle and decant (1 hour each, with a total cycle time of 4 hours, and 6 cycles per day). Influent to the system is directed sequentially to each of the four units for a fill period of one hour. That is, SBR #1 receives influent for 1 hour (while SBR #2 is decanting, SBR #3 is settling, and SBR #4 is reacting). At the end of the hour influent is directed to SBR #2 and each of the SBRs moves to the next stage in its cycle (SBR #1 starts the react period, SBR #3 starts decanting, and SBR #4 starts settling). In summary, each SBR is operated on the same cycle, but the cycles are offset from the adjacent unit by one hour.

Characteristics of the system:

SBR (full) volume	20 ML	
Depth	4.5 m	
Width	66 m	
Minimum decant level	50%	
Cycle:	Fill 1 hour	
	React 1 hour	
	Settle 1 hour	
	Decant 1 hour	
DO (fill and react phases)	2 mg/L	
Initial liquid hold-up	50%	
SBR underflow (for wastage):	20 ML/d from 1:45 to 2:00 hours in each cycle.	
Offsets:	**Unit**	**Offset (hours)**
	SBR #1	Zero
	SBR #2	3
	SBR #3	2
	SBR #4	1



The BioWin configuration



Example Album Chart

Single Continuous Flow SBR with Unaerated Mixed Prezone

The BioWin file for this system can be found in the **Data/Examples/SBR** directory under the name: **Continuous Flow SBR (mixed pre-zone).BWC**.

This single-SBR system is operated with continuous flow but intermittent decant. The SBR has an unaerated always-mixed prezone. The prezone is hydraulically connected to the main SBR zone; that is, the liquid levels in both zones are the same during operation. The SBR is operated on an 8-hour cycle, with a 6-hour aerated period in the main zone followed by 2 hours of settle/decant. Effluent is decanted over the last half hour of the cycle (from 7:30 to 8:00). The SBR main zone is aerated throughout the mixed period (fill-and-react period).

Influent feed is continuous with an average diurnal flow of 100 ML/d. The flow to the SBR continues during the settle-decant phase.

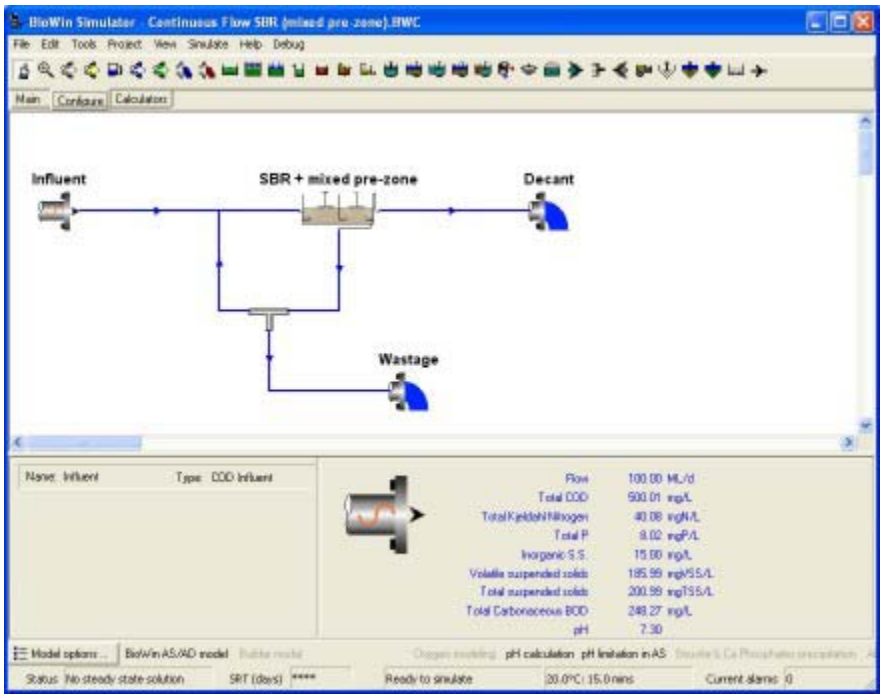
There is a 100 ML/d recycle from the SBR underflow to the always-mixed prezone (using the SBR underflow stream). This only operates during the 6-hour SBR mixed period, and is switched off when the SBR is in the settle-decant phase.

Sludge wastage to control SRT is via the side stream of a timed node splitter in the recycle stream. Wastage occurs at a rate of 48 ML/d over the last hour of the react period (5:00 to 6:00) immediately before settle/decant. Note that the wastage flow rate from the side stream of the splitter must be less than the recycle rate when wasting occurs.

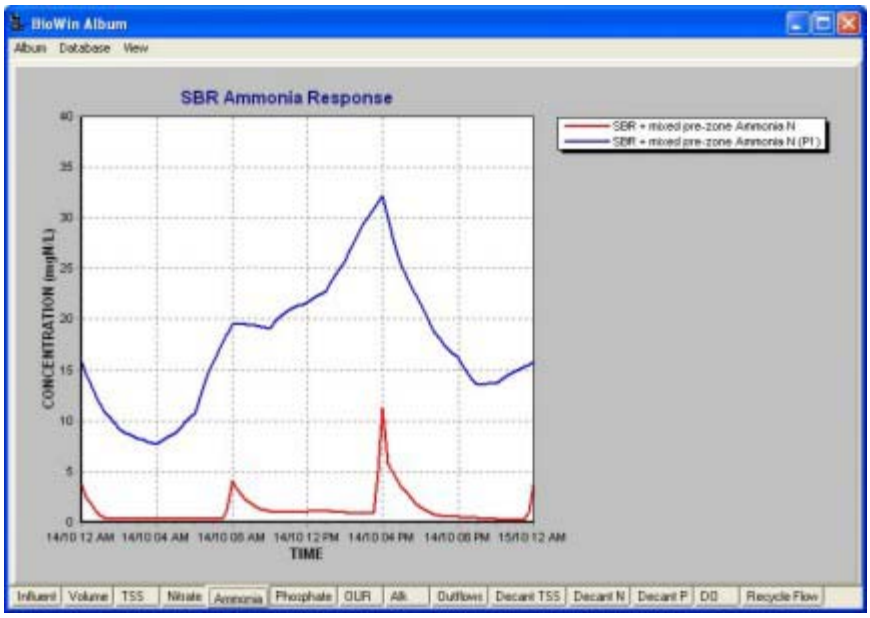
Characteristics of the system:

SBR main zone (full) volume	100 ML	
Prezone (full) volume	20 ML	

Depth	4.5 m	
Width	150 m	
Minimum decant level	50%	
Cycle:	Fill-and-react 6 hours	
	Settle 1.5 hours	
	Decant 0.5 hours	
DO (fill and react phases)	2 mg/L	
DO (prezone)	0 mg/L	
Initial liquid hold-up	50%	
Influent flow (average)	100 ML/d	
SBR underflow (for recycle):	100 ML/d from 0:00 to 6:00 hours in each cycle.	
Waste flow	48 ML/d from 5:00 to 6:00 hours in each cycle.	



The BioWin configuration



Example Album Chart

Single Continuous Flow SBR with Unaerated Settling Prezone

The BioWin file for this system can be found in the **Data/Examples/SBR** directory under the name: **Continuous Flow SBR (settle pre-zone).BWC**.

This single-SBR system is operated with continuous flow but intermittent decant. The SBR has an unaerated mix/settle prezone. The prezone is hydraulically connected to the main SBR zone; that is, the liquid levels in both zones are the same during operation. The SBR is operated on an 8-hour cycle, with a 6-hour aerated period in the main zone followed by 2 hours of settle/decant. Note that the prezone is in settling mode whenever the main SBR zone is in settling/decant mode. Effluent is decanted over the last half hour of the cycle (from 7:30 to 8:00). The SBR main zone is aerated throughout the mixed period (fill-and-react period).

Influent feed is continuous with an average diurnal flow of 100 ML/d. The flow to the SBR continues during the settle-decant phase.

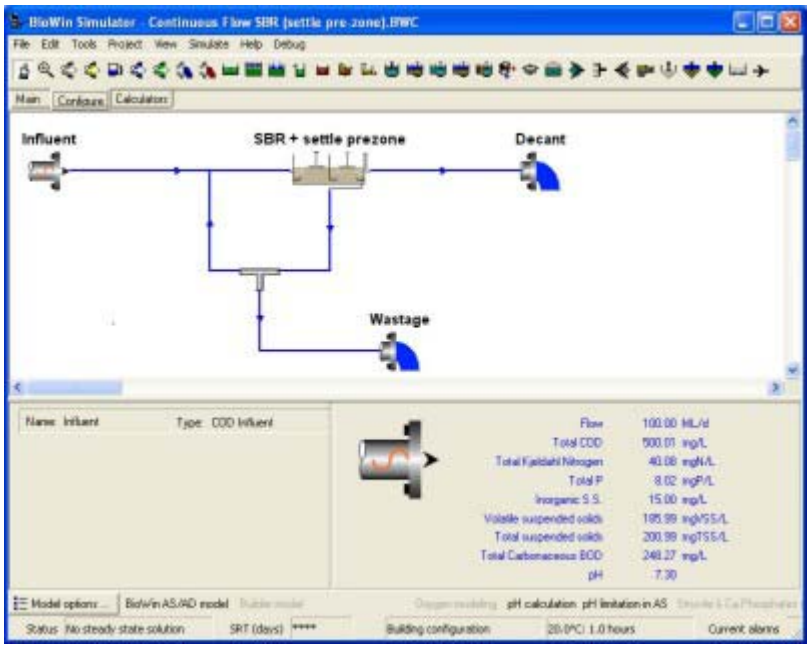
There is a 100 ML/d recycle from the SBR underflow to the mix/settle prezone (using the SBR underflow stream). This only operates during the 6-hour SBR mixed period, and is switched off when the SBR is in the settle-decant phase.

Sludge wastage to control SRT is via the side stream of a timed node splitter in the recycle stream. Wastage occurs at a rate of 48 ML/d over the last hour of the react period (5:00 to 6:00) immediately before settle/decant. Note that the wastage flow rate from the side stream of the splitter must be less than the recycle rate when wasting occurs.

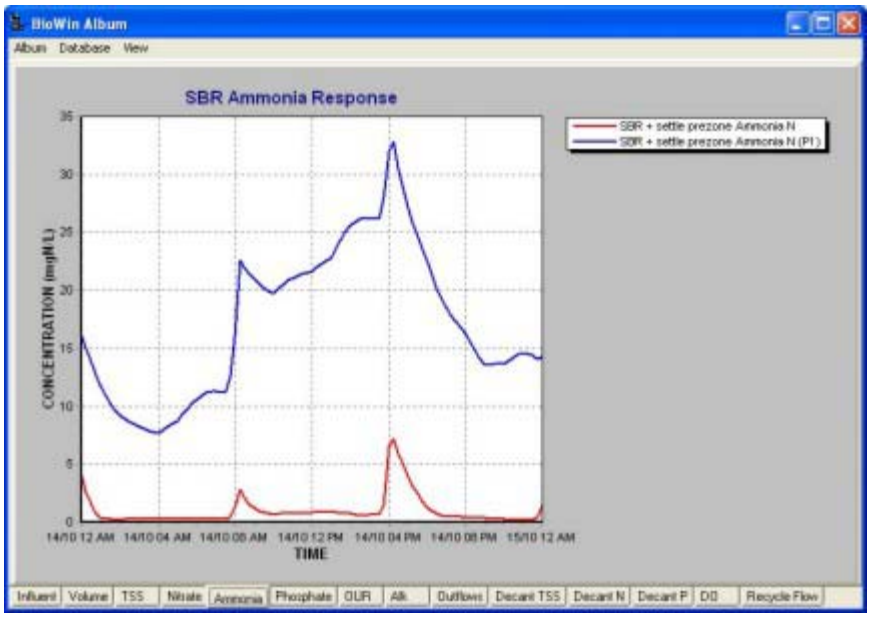
Characteristics of the system:

SBR main zone (full) volume	100 ML	
Prezone (full) volume	20 ML	

Depth	4.5 m	
Width	150 m	
Minimum decant level	50%	
Cycle:	Fill-and-react 6 hours	
	Settle 1.5 hours	
	Decant 0.5 hours	
DO (fill and react phases)	2 mg/L	
DO (prezone)	0 mg/L	
Initial liquid hold-up	50%	
Influent flow (average)	100 ML/d	
SBR underflow (for recycle):	100 ML/d from 0:00 to 6:00 hours in each cycle.	
Waste flow	48 ML/d from 5:00 to 6:00 hours in each cycle.	



The BioWin configuration



Example Album Chart

Single Continuous Flow SBR with Two Unaerated Mixed Prezones

The BioWin file for this system can be found in the **Data/Examples/SBR** directory under the name: **Continuous Flow SBR (two mixed pre-zones).BWC**.

This single-SBR system is operated with continuous flow but intermittent decant. The SBR has two unaerated always-mixed prezones. The prezones are hydraulically connected to the main SBR zone; that is, the liquid levels in all three zones are the same during operation. The SBR is operated on an 8-hour cycle, with a 6-hour aerated period in the main zone followed by 2 hours of settle/decant. Effluent is decanted over the last half hour of the cycle (from 7:30 to 8:00). The SBR main zone is aerated throughout the mixed period (fill-and-react period).

Influent feed is continuous with an average diurnal flow of 100 ML/d. The flow to the SBR continues during the settle-decant phase.

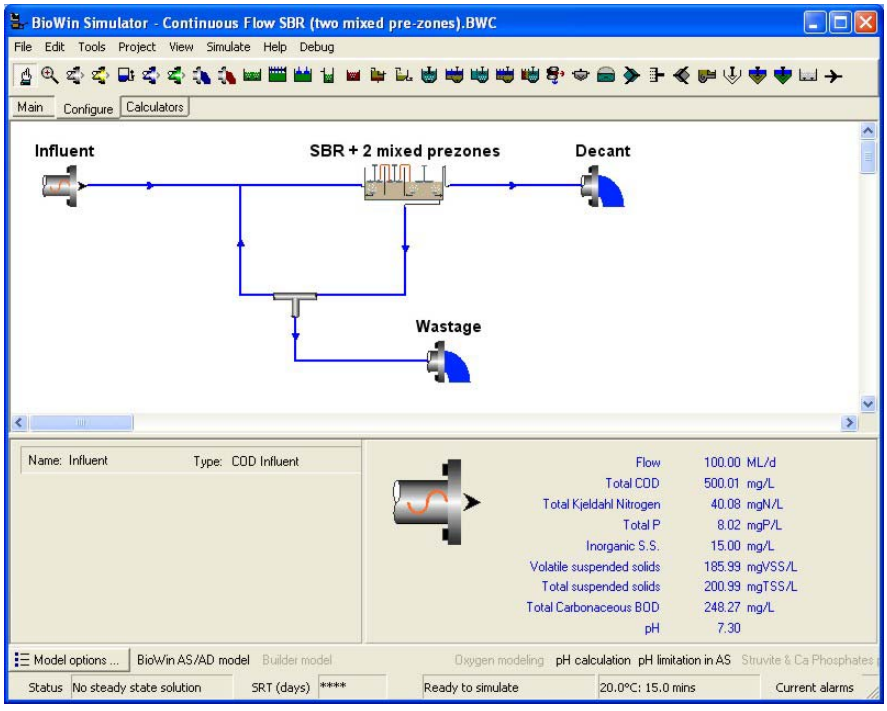
There is a 100 ML/d recycle from the SBR underflow to the first always-mixed prezone (using the SBR underflow stream). This only operates during the 6-hour SBR mixed period, and is switched off when the SBR is in the settle-decant phase.

Sludge wastage to control SRT is via the side stream of a timed node splitter in the recycle stream. Wastage occurs at a rate of 48 ML/d over the last hour of the react period (5:00 to 6:00) immediately before settle/decant. Note that the wastage flow rate from the side stream of the splitter must be less than the recycle rate when wasting occurs.

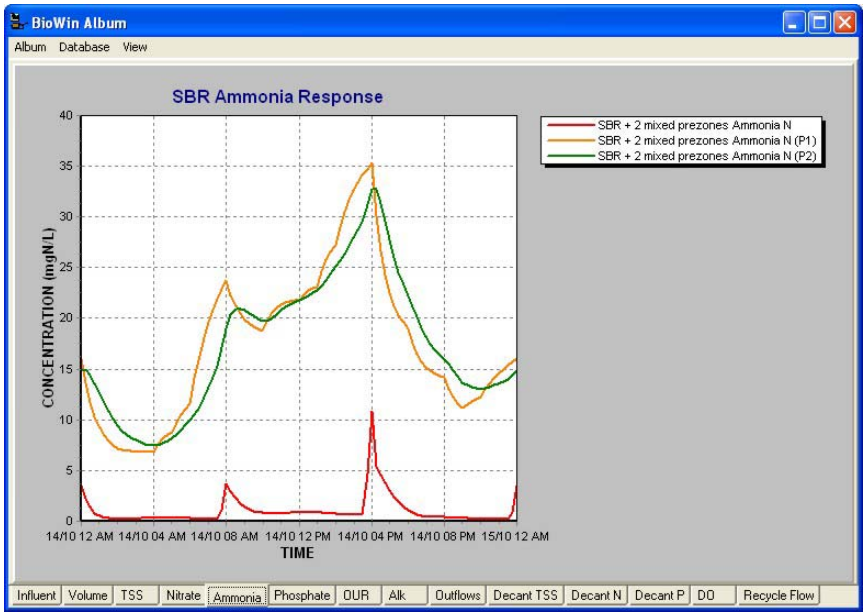
Characteristics of the system:

SBR main zone (full) volume	100 ML	
Prezone 1 (full) volume	10 ML	

Prezone 2 (full) volume	10 ML	
Depth	4.5 m	
Width	150 m	
Minimum decant level	50%	
Cycle:	Fill-and-react 6 hours	
	Settle 1.5 hours	
	Decant 0.5 hours	
DO (fill and react phases)	2 mg/L	
DO (prezones)	0 mg/L	
Initial liquid hold-up	50%	
Influent flow (average)	100 ML/d	
SBR underflow (for recycle):	100 ML/d from 0:00 to 6:00 hours in each cycle.	
Waste flow	48 ML/d from 5:00 to 6:00 hours in each cycle.	



The BioWin configuration



Example Album Chart

Single Continuous Flow SBR with Two Unaerated Settling Prezones

The BioWin file for this system can be found in the **Data/Examples/SBR** directory under the name: **Continuous Flow SBR (two settle pre-zones).BWC**.

This single-SBR system is operated with continuous flow but intermittent decant. The SBR has two unaerated mix/settle prezones. The prezones are hydraulically connected to the main SBR zone; that is, the liquid levels in all three zones are the same during operation. The SBR is operated on an 8-hour cycle, with a 6-hour aerated period in the main zone followed by 2 hours of settle/decant. Note that the prezones are in settling mode whenever the main SBR zone is in settling/decant mode. Effluent is decanted over the last half hour of the cycle (from 7:30 to 8:00). The SBR main zone is aerated throughout the mixed period (fill-and-react period).

Influent feed is continuous with an average diurnal flow of 100 ML/d. The flow to the SBR continues during the settle-decant phase.

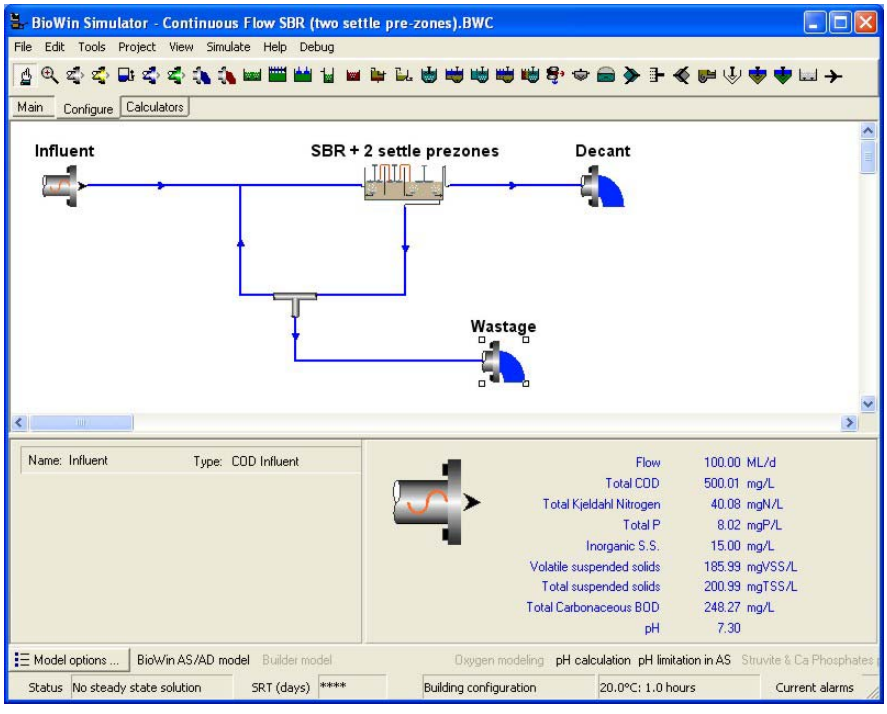
There is a 100 ML/d recycle from the SBR underflow to the first mix/settle prezone (using the SBR underflow stream). This only operates during the 6-hour SBR mixed period, and is switched off when the SBR is in the settle-decant phase.

Sludge wastage to control SRT is via the side stream of a timed node splitter in the recycle stream. Wastage occurs at a rate of 48 ML/d over the last hour of the react period (5:00 to 6:00) immediately before settle/decant. Note that the wastage flow rate from the side stream of the splitter must be less than the recycle rate when wasting occurs.

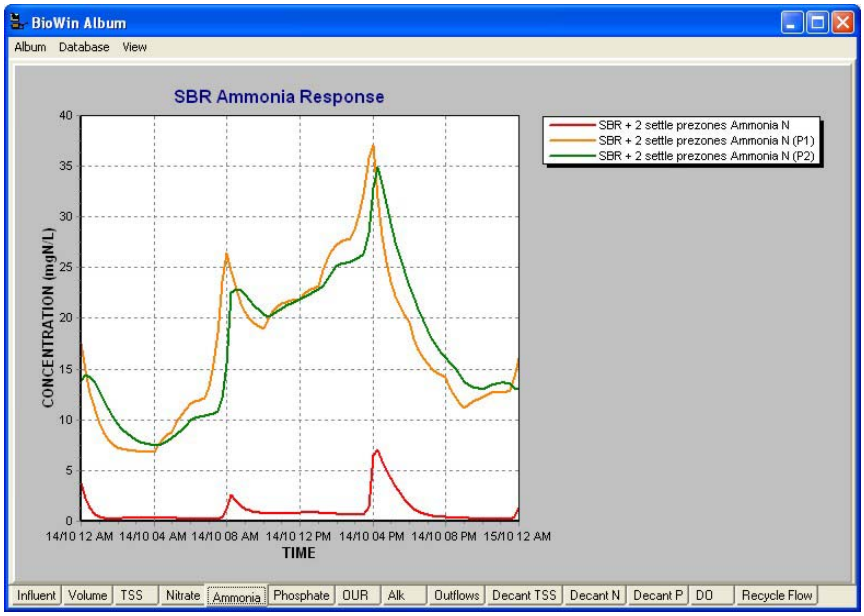
Characteristics of the system:

SBR main zone (full) volume	100 ML	
Prezone 1 (full)	10 ML	

volume		
Prezone 2 (full) volume	10 ML	
Depth	4.5 m	
Width	150 m	
Minimum decant level	50%	
Cycle:	Fill-and-react 6 hours	
	Settle 1.5 hours	
	Decant 0.5 hours	
DO (fill and react phases)	2 mg/L	
DO (prezones)	0 mg/L	
Initial liquid hold-up	50%	
Influent flow (average)	100 ML/d	
SBR underflow (for recycle):	100 ML/d from 0:00 to 6:00 hours in each cycle.	
Waste flow	48 ML/d from 5:00 to 6:00 hours in each cycle.	



The BioWin configuration



Example Album Chart

Two-in-Parallel SBRs with Unaerated Mixed Prezone

The BioWin file for this system can be found in the **Data/Examples/SBR** directory under the name: **Two SBRs (mixed pre-zone).BWC**.

This system has two SBRs in parallel, each with an unaerated always-mixed prezone. The prezone is hydraulically connected to the main SBR zone; that is, the liquid level in both zones are the same during operation. The SBRs are each operated on an 8-hour cycle, with a 6-hour aerated period in the main zone followed by 2 hours of settle/decant. Effluent is decanted over the last half hour of the cycle (from 7:30 to 8:00). Each SBR is aerated throughout the mixed period (fill-and-react period).

Influent feed is continuous with an average diurnal flow of 100 ML/d. The flow is routed to each train for 4 hour intervals (and flow is displaced to the downstream SBR). Flow does not pass to the SBRs during the settle-decant phase. The cycle for the second SBR has an offset of 4 hours.

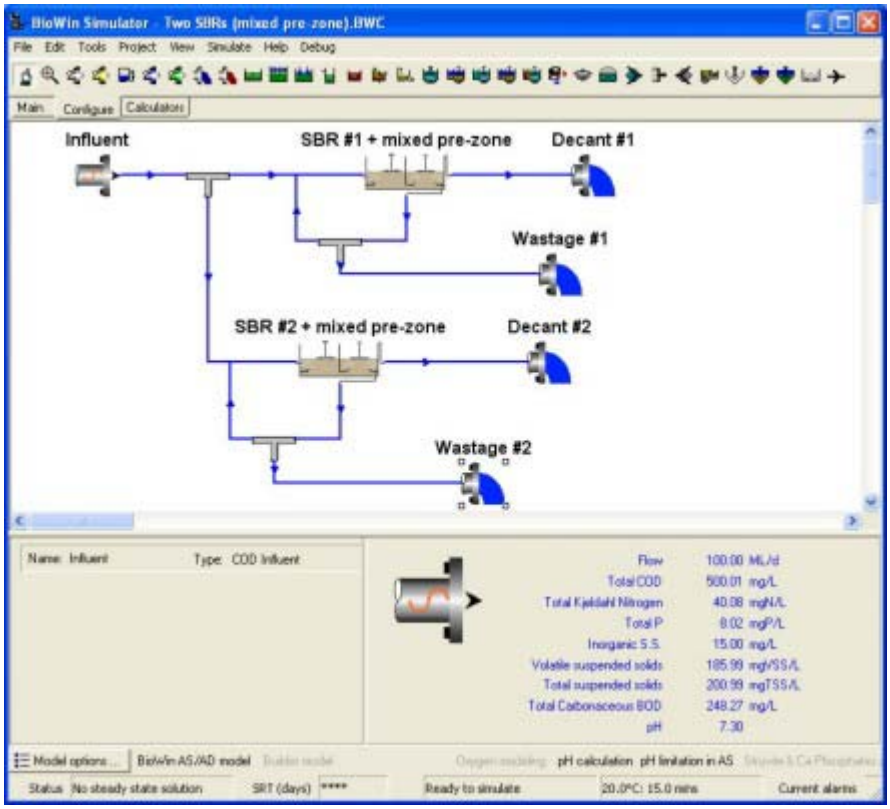
There is a 100 ML/d recycle from the SBR underflow to the always-mixed prezone (using the SBR underflow stream). This only operates during the 6-hour SBR mixed period, and is switched off when the SBR is in the settle-decant phase.

Sludge wastage to control SRT is via the side stream of a timed node splitter in the recycle stream. Wastage occurs at a rate of 24 ML/d over the last hour of the react period (5:00 to 6:00) immediately before settle/decant. Note that the wastage flow rate from the side stream of the splitter must be less than the recycle rate when wasting occurs.

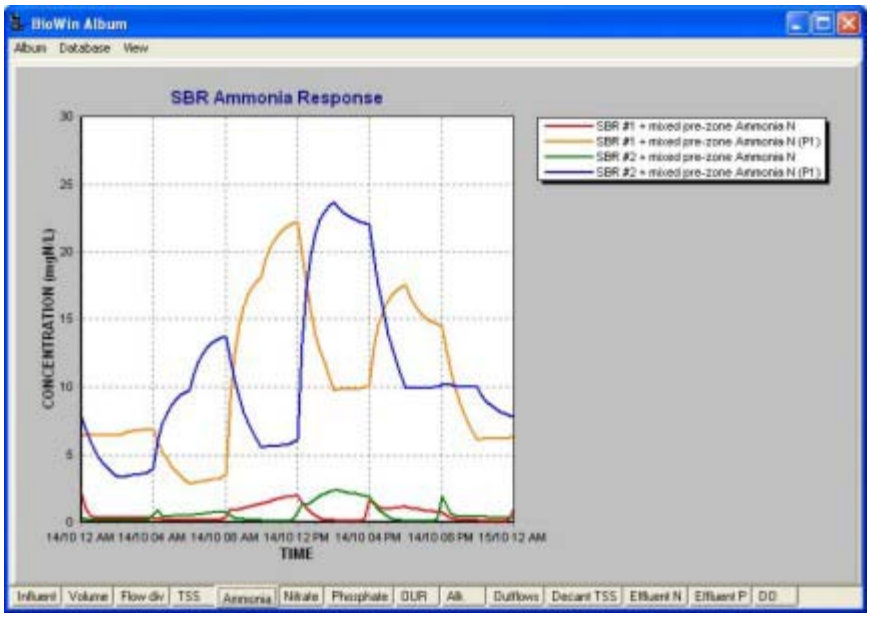
Characteristics of the system:

SBR main zone (full) volume	50 ML	
Prezone (full) volume	10 ML	

Depth	4.5 m	
Width	105 m	
Minimum decant level	50%	
Cycle:	Fill 4 hours	
	React 2 hours	
	Settle 1.5 hours	
	Decant 0.5 hours	
DO (fill and react phases)	2 mg/L	
DO (prezone)	0 mg/L	
Initial liquid hold-up	50%	
Influent flow (average)	100 ML/d	
SBR underflow (for recycle):	100 ML/d from 0:00 to 6:00 hours in each cycle.	
Waste flow	24 ML/d from 5:00 to 6:00 hours in each cycle.	
Offsets:	**Unit**	**Offset (hours)**
	SBR #1	Zero
	SBR #2	4



The BioWin configuration



Example Album Chart

Two-in-Parallel SBRs with Unaerated Settling Prezone

The BioWin file for this system can be found in the **Data/Examples/SBR** directory under the name: **Two SBRs (settle pre-zone).BWC**.

This system has two SBRs in parallel, each with an unaerated mix/settle prezone. The prezone is hydraulically connected to the main SBR zone; that is, the liquid level in both zones are the same during operation. The SBRs are each operated on an 8-hour cycle, with a 6-hour aerated period in the main zone followed by 2 hours of settle/decant. Note that the prezone is in settling mode whenever the main SBR zone is in settling/decant mode. Effluent is decanted over the last half hour of the cycle (from 7:30 to 8:00). Each SBR is aerated throughout the mixed period (fill-and-react period).

Influent feed is continuous with an average diurnal flow of 100 ML/d. The flow is routed to each train for 4 hour intervals (and flow is displaced to the downstream SBR). Flow does not pass to the SBRs during the settle-decant phase. The cycle for the second SBR has an offset of 4 hours.

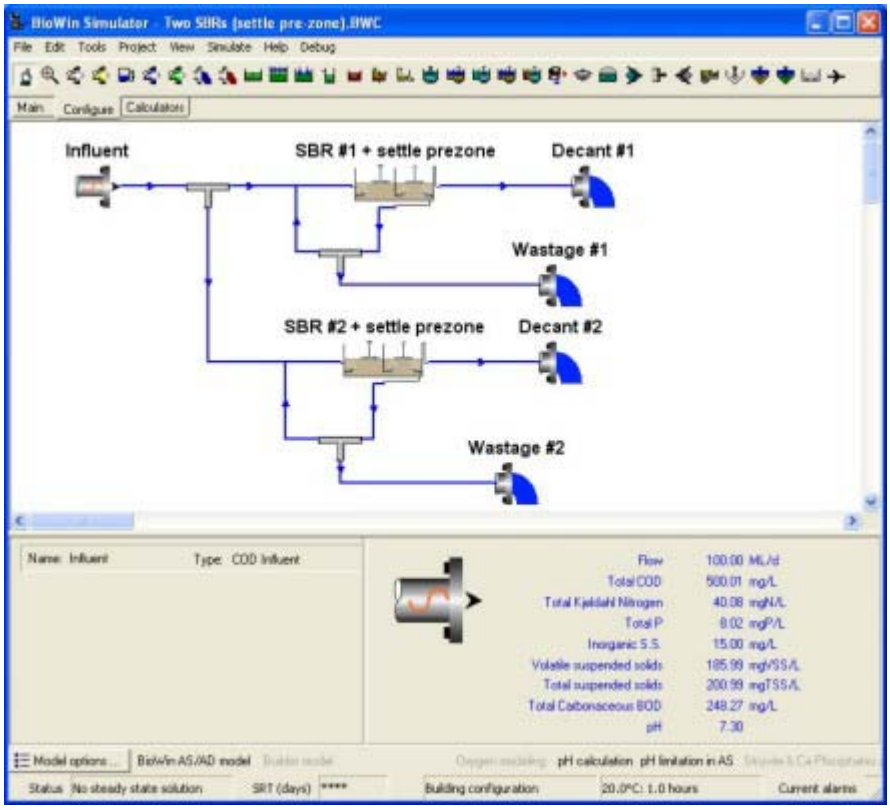
There is a 100 ML/d recycle from the SBR underflow to the mix/settle prezone (using the SBR underflow stream). This only operates during the 6-hour SBR mixed period, and is switched off when the SBR is in the settle-decant phase.

Sludge wastage to control SRT is via the side stream of a timed node splitter in the recycle stream. Wastage occurs at a rate of 24 ML/d over the last hour of the react period (5:00 to 6:00) immediately before settle/decant. Note that the wastage flow rate from the side stream of the splitter must be less than the recycle rate when wasting occurs.

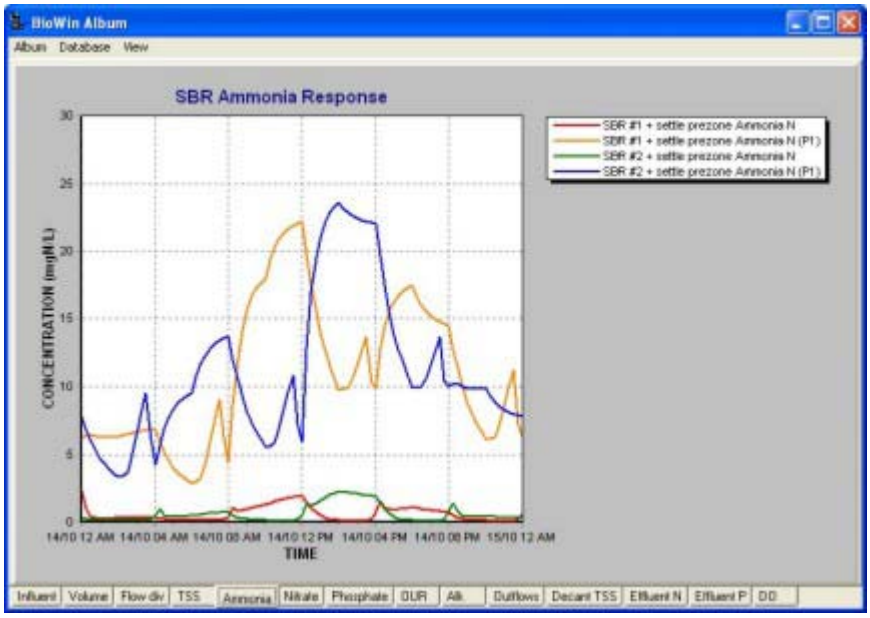
Characteristics of the system:

SBR main zone (full) volume	50 ML	

Prezone (full) volume	10 ML	
Depth	4.5 m	
Width	105 m	
Minimum decant level	50%	
Cycle:	Fill 4 hours	
	React 2 hours	
	Settle 1.5 hours	
	Decant 0.5 hours	
DO (fill and react phases)	2 mg/L	
DO (prezone)	0 mg/L	
Initial liquid hold-up	50%	
Influent flow (average)	100 ML/d	
SBR underflow (for recycle):	100 ML/d from 0:00 to 6:00 hours in each cycle.	
Waste flow	24 ML/d from 5:00 to 6:00 hours in each cycle.	
Offsets:	**Unit**	**Offset (hours)**
	SBR #1	Zero
	SBR #2	4



The BioWin configuration



Example Album Chart

Two-in-Parallel SBRs with Two Unaerated Mixed Prezones

The BioWin file for this system can be found in the **Data/Examples/SBR** directory under the name: **Two SBRs (two mixed pre-zones).BWC**.

This system has two SBRs in parallel, each with two unaerated always-mixed prezones. The prezones are hydraulically connected to one another and to the main SBR zone; that is, the liquid levels in all three zones are the same during operation. The SBRs are each operated on an 8-hour cycle, with a 6-hour aerated period in the main zone followed by 2 hours of settle/decant. Effluent is decanted over the last half hour of the cycle (from 7:30 to 8:00). Each SBR is aerated throughout the mixed period (fill-and-react period).

Influent feed is continuous with an average diurnal flow of 100 ML/d. The flow is routed to each train for 4 hour intervals (and flow is displaced to the downstream SBR). Flow does not pass to the SBRs during the settle-decant phase. The cycle for the second SBR has an offset of 4 hours.

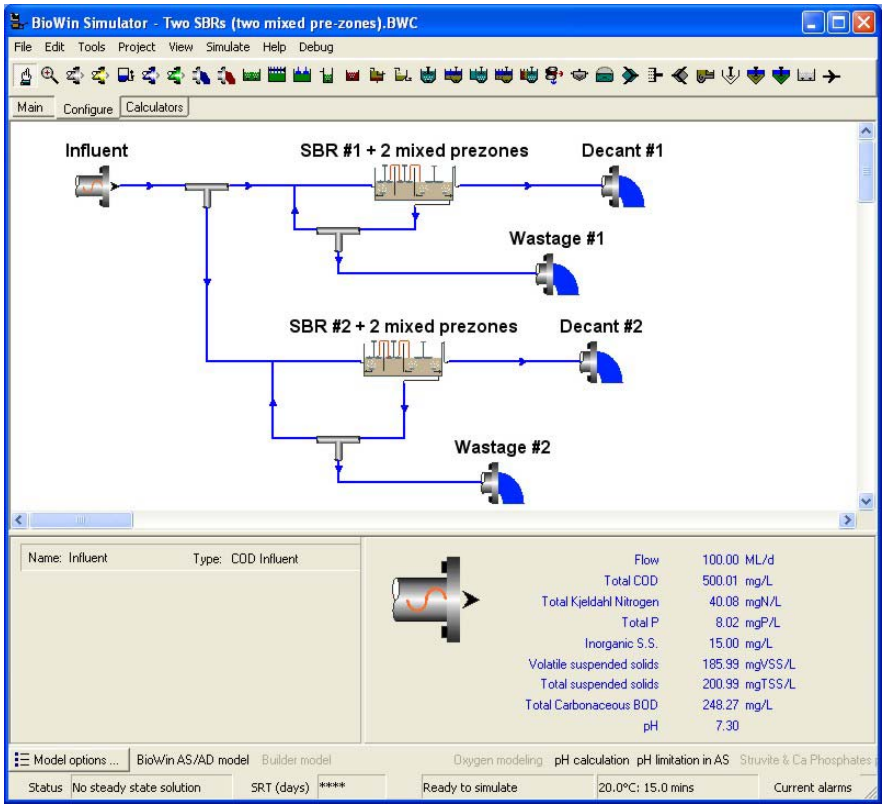
There is a 100 ML/d recycle from the SBR underflow to the first always-mixed prezone (using the SBR underflow stream). This only operates during the 6-hour SBR mixed period, and is switched off when the SBR is in the settle-decant phase.

Sludge wastage to control SRT is via the side stream of a timed node splitter in the recycle stream. Wastage occurs at a rate of 24 ML/d over the last hour of the react period (5:00 to 6:00) immediately before settle/decant. Note that the wastage flow rate from the side stream of the splitter must be less than the recycle rate when wasting occurs.

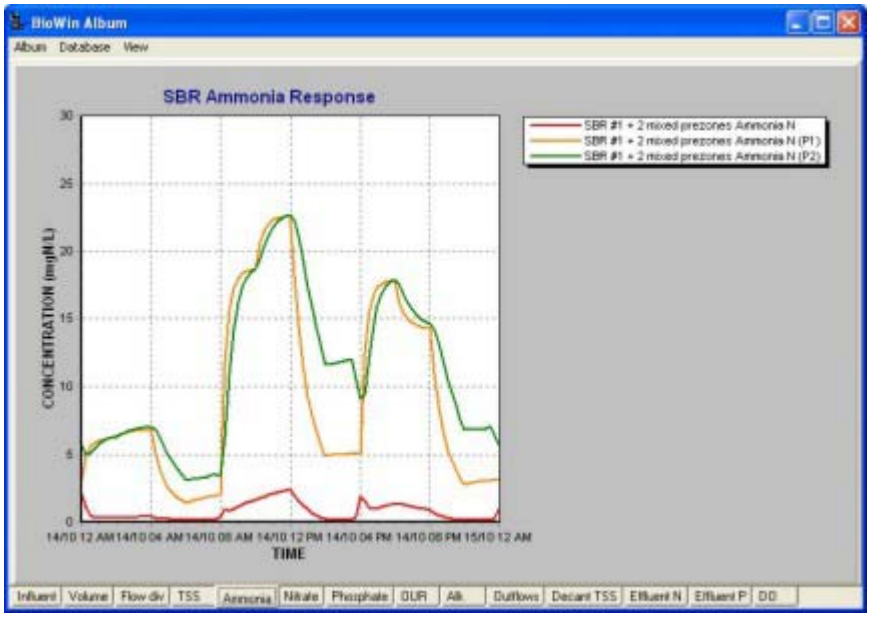
Characteristics of the system:

SBR main zone (full) volume	50 ML	
Prezone 1 (full) volume	5 ML	
Prezone 2 (full) volume	5 ML	
Depth	4.5 m	
Width	105 m	
Minimum decant level	50%	
Cycle:	Fill 4 hours	
	React 2 hours	
	Settle 1.5 hours	
	Decant 0.5 hours	
DO (fill and react phases)	2 mg/L	
DO (prezones)	0 mg/L	
Initial liquid hold-up	50%	
Influent flow (average)	100 ML/d	
SBR underflow (for recycle):	100 ML/d from 0:00 to 6:00 hours in each cycle.	
Waste flow	24 ML/d from 5:00 to 6:00 hours in each cycle.	

Offsets:	Unit	Offset (hours)
	SBR #1	Zero
	SBR #2	4



The BioWin configuration



Example Album Chart

Two-in-Parallel SBRs with Two Unaerated Settling Prezones

The BioWin file for this system can be found in the **Data/Examples/SBR** directory under the name: **Two SBRs (two settle pre-zones).BWC**.

This system has two SBRs in parallel, each with two unaerated mix/settle prezones. The prezones are hydraulically connected to one another and to the main SBR zone; that is, the liquid levels in all three zones are the same during operation. The SBRs are each operated on an 8-hour cycle, with a 6-hour aerated period in the main zone followed by 2 hours of settle/decant. Note that the prezones are both in settling mode whenever the main SBR zone is in settling/decant mode. Effluent is decanted over the last half hour of the cycle (from 7:30 to 8:00). Each SBR is aerated throughout the mixed period (fill-and-react period).

Influent feed is continuous with an average diurnal flow of 100 ML/d. The flow is routed to each train for 4 hour intervals (and flow is displaced to the downstream SBR). Flow does not pass to the SBRs during the settle-decant phase. The cycle for the second SBR has an offset of 4 hours.

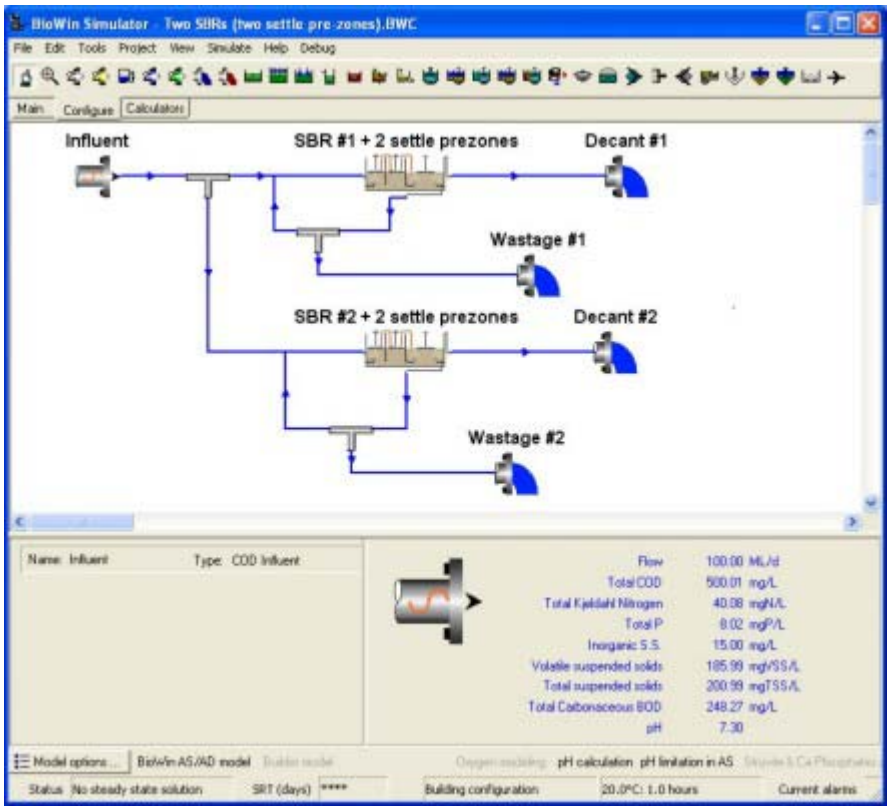
There is a 100 ML/d recycle from the SBR underflow to the first mix/settle prezone (using the SBR underflow stream). This only operates during the 6-hour SBR mixed period, and is switched off when the SBR is in the settle-decant phase.

Sludge wastage to control SRT is via the side stream of a timed node splitter in the recycle stream. Wastage occurs at a rate of 24 ML/d over the last hour of the react period (5:00 to 6:00) immediately before settle/decant. Note that the wastage flow rate from the side stream of the splitter must be less than the recycle rate when wasting occurs.

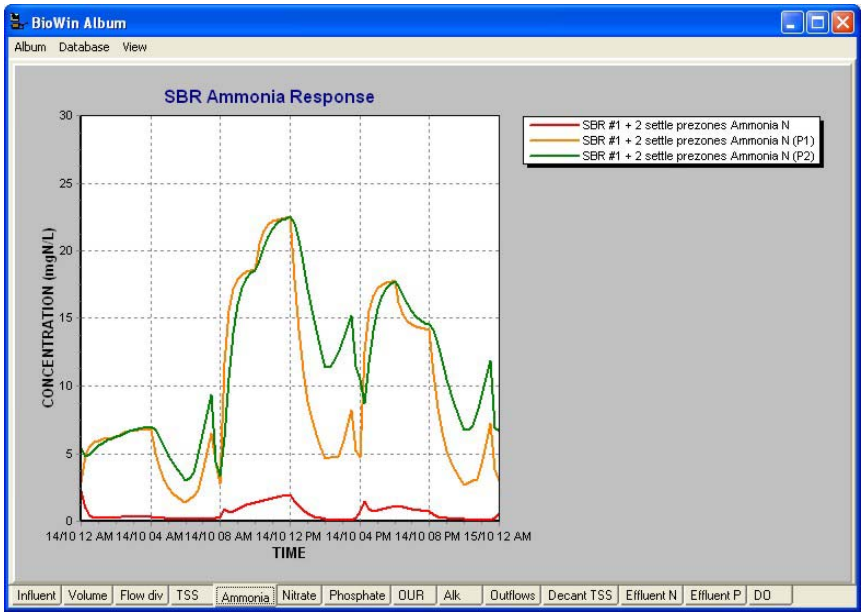
Characteristics of the system:

SBR main zone (full) volume	50 ML	
Prezone 1 (full) volume	5 ML	
Prezone 2 (full) volume	5 ML	
Depth	4.5 m	
Width	105 m	
Minimum decant level	50%	
Cycle:	Fill 4 hours	
	React 2 hours	
	Settle 1.5 hours	
	Decant 0.5 hours	
DO (fill and react phases)	2 mg/L	
DO (prezone)	0 mg/L	
Initial liquid hold-up	50%	

Influent flow (average)	100 ML/d	
SBR underflow (for recycle):	100 ML/d from 0:00 to 6:00 hours in each cycle.	
Waste flow	24 ML/d from 5:00 to 6:00 hours in each cycle.	
Offsets:	**Unit**	**Offset (hours)**
	SBR #1	Zero
	SBR #2	4



The BioWin configuration



Example Album Chart

Two-in-Parallel SBRs with Unaerated Constant-volume Prezone

The BioWin file for this system can be found in the **Data/Examples/SBR** directory under the name: **Two SBRs (constant volume pre-zone).BWC**.

This system has two SBRs in parallel, each preceded by a constant-volume unaerated reactor. Note that the volume of each unaerated reactor is fixed; that is, it is not hydraulically connected to the SBR and its volume does not change during operation. The SBRs are each operated on an 8-hour cycle, with a 6-hour aerated period followed by 2 hours of settle/decant. Effluent is decanted over the last half hour of the cycle (from 7:30 to 8:00). Each SBR is aerated throughout the mixed period (fill-and-react period).

Influent feed is continuous with an average diurnal flow of 100 ML/d. The flow is routed to each train for 4 hour intervals (and flow is displaced to the downstream SBR). Flow does not pass to the SBRs during the settle-decant phase. The cycle for the second SBR has an offset of 4 hours.

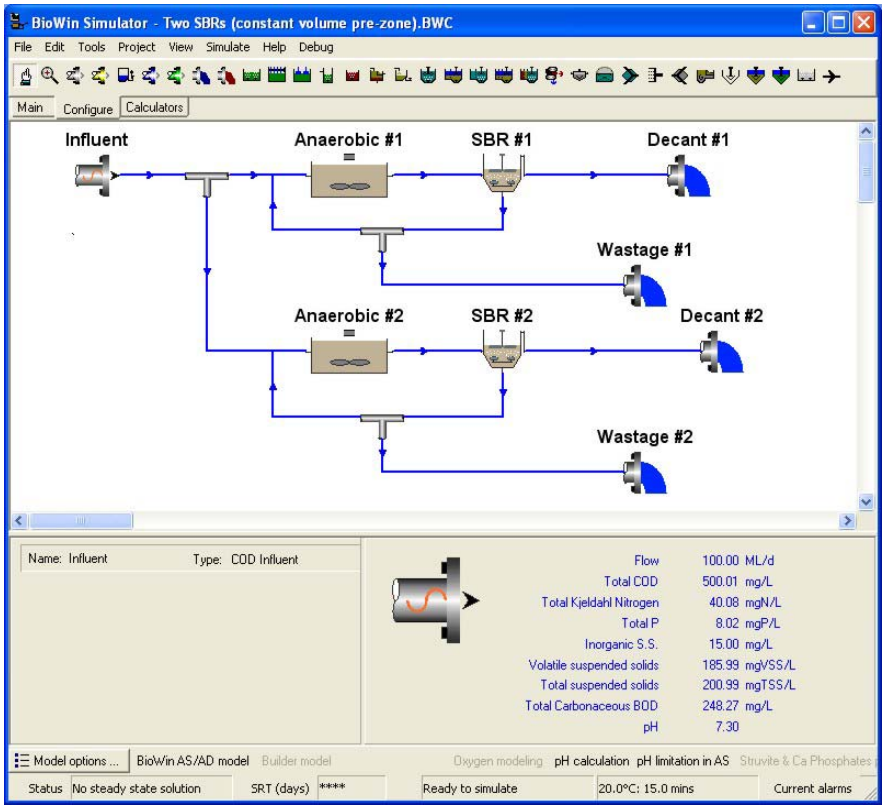
There is a 100 ML/d recycle from the SBR underflow to the unaerated reactor (using the SBR underflow stream). This only operates during the 6-hour SBR mixed period, and is switched off when the SBR is in the settle-decant phase.

Sludge wastage to control SRT is via the side stream of a timed node splitter in the recycle stream. Wastage occurs at a rate of 24 ML/d over the last hour of the react period (5:00 to 6:00) immediately before settle/decant. Note that the wastage flow rate from the side stream of the splitter must be less than the recycle rate when wasting occurs.

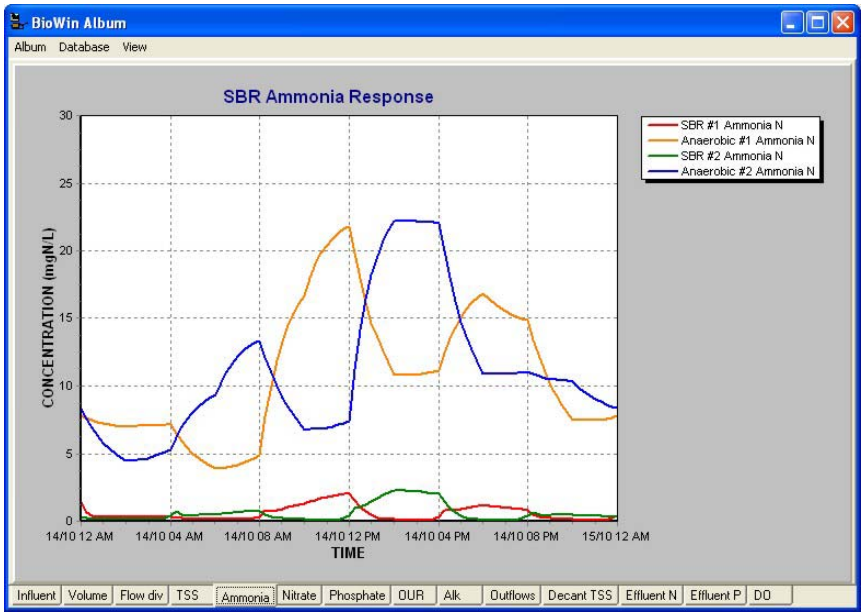
Characteristics of the system:

SBR (full) volume	50 ML	
Anaerobic volume (constant)	10 ML	

Depth	4.5 m	
Width	105 m	
Minimum decant level	50%	
Cycle:	Fill 4 hours	
	React 2 hours	
	Settle 1.5 hours	
	Decant 0.5 hours	
DO (fill and react phases)	2 mg/L	
DO (anaerobic)	0 mg/L	
Initial liquid hold-up	50%	
Influent flow (average)	100 ML/d	
SBR underflow (for recycle):	100 ML/d from 0:00 to 6:00 hours in each cycle.	
Waste flow	24 ML/d from 5:00 to 6:00 hours in each cycle.	
Offsets:	**Unit**	**Offset (hours)**
	SBR #1	Zero
	SBR #2	4



The BioWin configuration



Example Album Chart

Two-in-Parallel Single-Tank SBRs with Unaerated/Aerated Periods

The BioWin file for this system can be found in the **Data/Examples/SBR** directory under the name: **Two SBRs (single tank BNR).bwc**.

This system has two SBRs in parallel. The SBRs are each operated on an 8-hour cycle, with a 6-hour fill-and-react period followed by 2 hours of settle/decant. Effluent is decanted over the last half hour of the cycle (from 7:30 to 8:00). Each SBR is unaerated for the first 2 hours of the mixed period (fill-and-react period) and then is aerated until the start of settle-decant.

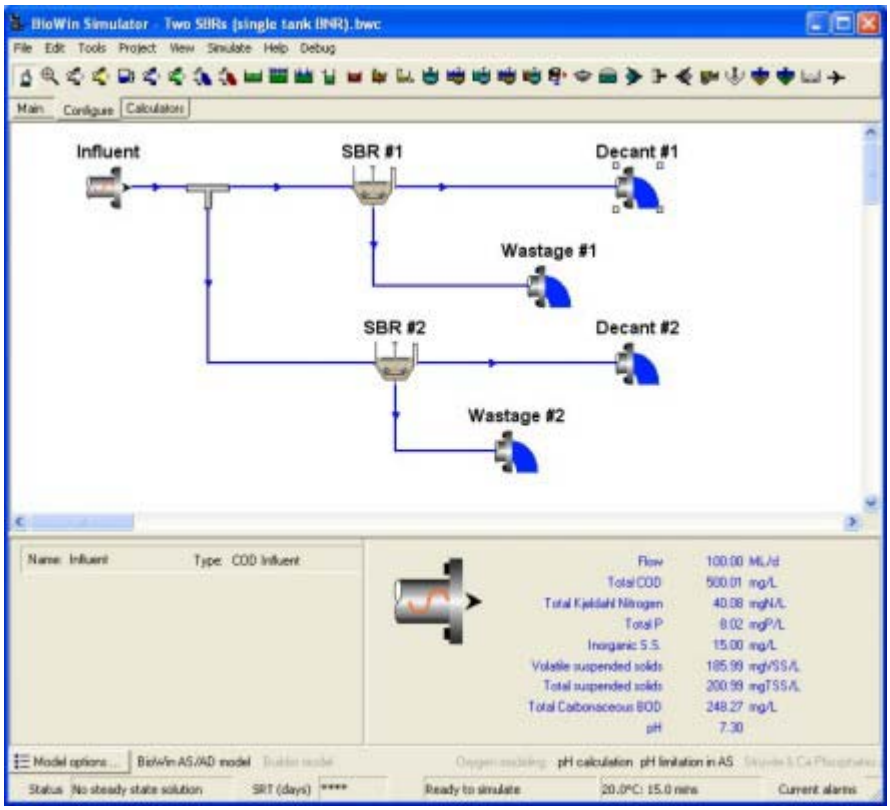
Influent feed is continuous with an average diurnal flow of 100 ML/d. The flow is routed to each train for 4 hour intervals. Flow does not pass to the SBRs during the settle-decant phase. The cycle for the second SBR has an offset of 4 hours.

Sludge wastage to control SRT is via the SBR underflow. Wastage occurs at a rate of 24 ML/d over the last hour of the react period (5:00 to 6:00) immediately before settle/decant.

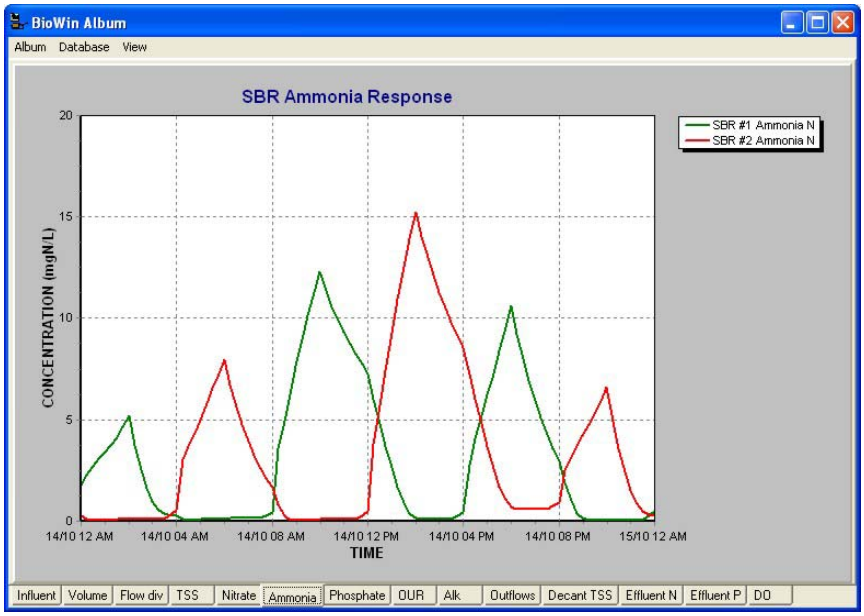
Characteristics of the system:

SBR (full) volume	50 ML	
Depth	4.5 m	
Width	105 m	
Minimum decant level	50%	
Cycle:	Fill 4 hours	
	React 2 hours	
	Settle 1.5 hours	
	Decant 0.5 hours	

DO (0:00 to 2:00 hours)	0 mg/L	
DO (2:00 to 6:00 hours)	2 mg/L	
Initial liquid hold-up	50%	
Influent flow (average)	100 ML/d	
SBR underflow (for waste):	24 ML/d from 5:00 to 6:00 hours in each cycle.	
Offsets:	**Unit**	**Offset (hours)**
	SBR #1	Zero
	SBR #2	4



The BioWin configuration



Example Album Chart

Wastewater Characterization Fill-and-Draw SBR

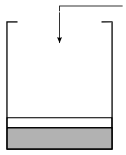
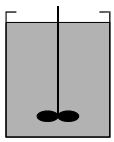
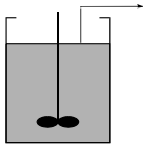
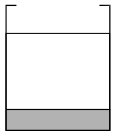
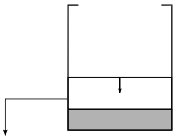
The BioWin file for this system can be found in the **Data/Examples/SBR** directory under the name: **Fill-and-Draw SBR.BWC**.

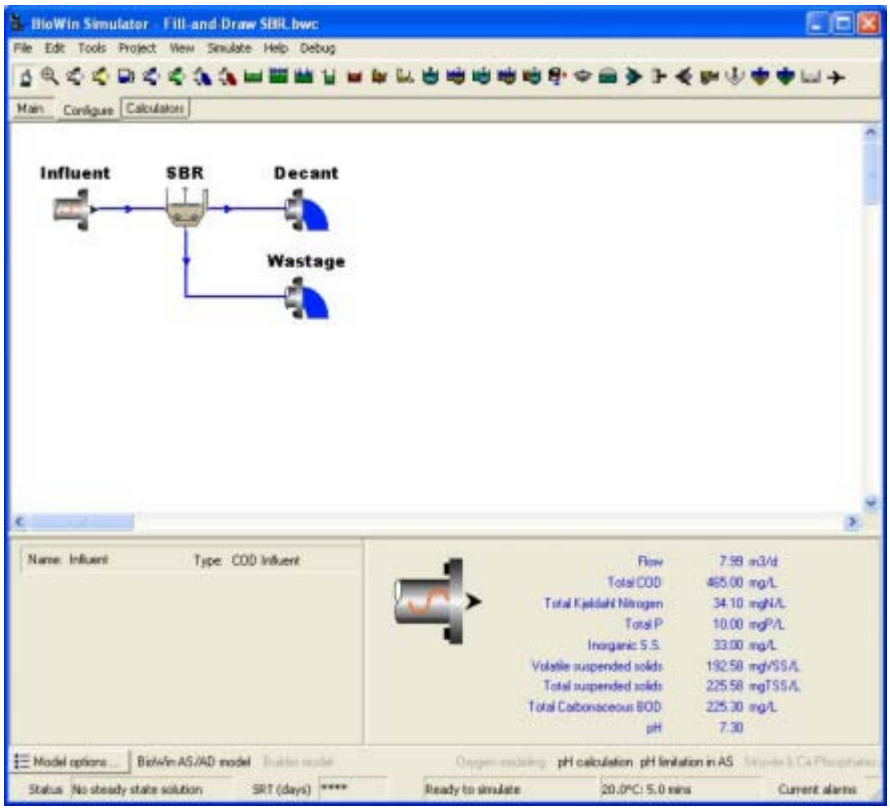
This example illustrates the operation of a fill-and-draw SBR used in a wastewater characterization exercise.

The SBR is operated according to the sequence outlined in the table and chart below. The volume of the SBR is 10.5 m^3 but only reaches a liquid hold-up of 10.1 m^3.

V_P = 10.1 m^3	**Temperature = 20°C**
V_{WW} = 8 m^3	θ_X = 10.1/0.67 = 15 days
q_W = 0.67 m^3/d	
Wastewater Parameter	**Value**
S_{TI}	465 mg COD/L
N_{TI}	34.1 mg N/L
ISS	33 mg/L
f_{BS}	0.11
f_{US}	0.05
f_{UP}	0.13
f_{NA}	0.48

PHASE	TIME	REACTOR CONDITION	ACTION

FILL	T = 0 (Instantaneous fill)		V_{ww} of wastewater is added to reactor containing V_p - V_{ww} of mixed liquor, i.e. reactor is filled to the maximum volume (V_p).
REACT	T = 0 – 23:15 hours		Reactor volume constant (V_p). Mixing and aeration on.
WASTE	T = 23:05 – 23:15 hours (Instantaneous wastage)		Mixing and aeration on. Withdraw volume of mixed liquor for wastage (q_w). $q_w = V_p/\theta_x$.
SETTLE	T = 23:15 - 24 hours		No mixing or aeration. Allow sludge to settle.
DRAW	T = 23:40 – 24 hours (Instantaneous draw of treated water)		No mixing or aeration. Decant off supernatant (effluent) volume of (V_{ww} - q_w), leaving a volume of V_p – V_{ww} in the reactor.



Configuring the system

1. Set up the SBR configuration shown above. Double-click the SBR element, and on the **Dimensions** tab, enter the appropriate volumes. Set the minimum decant level as 20 %. Click **OK** to close the dialog box.
2. Influent flow rate: We wish to fill the reactor with the 8 m^3 of influent over 5 minutes, starting at time zero. The influent flow rate should be at least:

$$\frac{8\ m^3}{5\ min} \cdot \frac{60\ min}{1\ hr} \cdot \frac{24\ hr}{1\ day} = 2304\ m^3/d$$

3. Double-click the influent element and specify the **Influent type** as variable. Click the **Edit data** button to enter the influent characteristics and flows.
4. Ensure that the **Cycle time** is set to 1 day.
5. Below the cells for entering data, specify the **Time in grid** as minutes. Ensure that the **Flow units** are in m^3/d.
   - The first row begins at time zero minutes. Enter the flow calculated above in the flow column. Enter the appropriate data in the various columns (i.e. COD, TKN, etc.).
   - After five minutes, we want to turn the flow off. In the second row under the time column, type 5. In the second row flow column, type 0. You don't have to worry about entering wastewater data for the second row, so click **Close** to exit.
6. Cycle operation: Double-click the SBR element, and on the **Operation** tab, specify the cycle time as 24 hours (zero offset). Mix the SBR from the start of the cycle until 23h15m. Specify a constant DO setpoint of 3 mg/L. Decant to the minimum level from 23h40m until the end of the cycle.
7. Wastage: We wish to waste a mixed liquor volume of 0.67 m^3 over the last 10 minutes of the mixed period, i.e. from 23h05m to 23h15m (1385 – 1395 minutes). The wastage rate over this period should be:

$$\frac{0.67\ m^3}{10\ min} \cdot \frac{60\ min}{1\ hr} \cdot \frac{24\ hr}{1\ day} = 96.48\ m^3/d$$

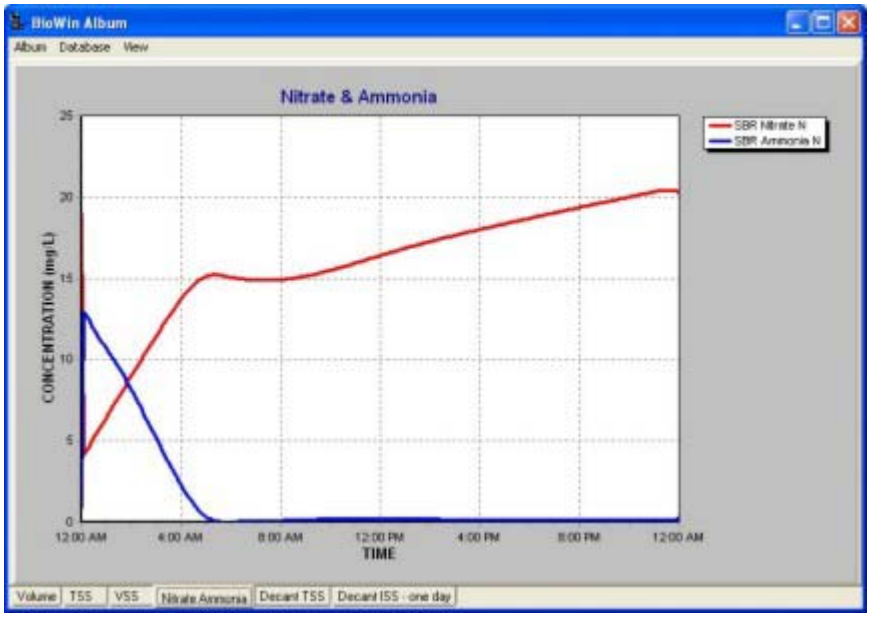
   - On the SBR **Underflow** tab, select **Flow pattern** and click the **Pattern…** button. Ensure that the **Time in** grid is in minutes. In the first row of the itinerary, at time zero, enter 0 for the underflow. In the second row, enter 1385 for the time and 96.48 for the flow. In the third row, enter 1395 for the time and 0 for the flow.

Setting up charts and running simulations

1. Open the album and set up charts for SBR TSS and Volume.
2. In the main window **Project|Database** sub-menu, select **Data interval…** and set the monitoring interval to 5 minutes. Often things

are changing rapidly in SBR systems, so a short data interval is necessary to track the changes.

3. Begin a dynamic simulation and simulate for 20 days, starting from seed concentrations.
4. Switch to the album and view the SBR volume response carefully. This often provides a check that you have set up the system correctly.
5. View the TSS response. As the simulation moves towards 20 days the TSS should "settle down" in the SBR. Likely, after 20 days you will not yet have attained a steady state. If so, continue the simulation until 40 or 50 days as appropriate.
6. If the system has yet to stabilize by 50 days, stop the simulation and restart the simulation for 20 days from **Current conditions**. This sets the simulation clock back to zero, but you will be viewing the system response from Day 50 on.
7. Continue simulation until you are satisfied that the system has attained a quasi "steady state". Then Cancel the simulation and save the BioWin file.
8. In the album, set up charts for VSS, ammonia and nitrate concentrations, and OUR. Re-start the simulation for a 24 hour period, and examine the responses.
9. Use the main window **Project|Parameters|Kinetic…** menu command to open the **Model Parameter Editor** and change the nitrification rate. View the response.



Example Album Chart

Note: If you change a parameter such as f_{UP} which affects sludge production you must simulate for an extended period to allow the system to stabilize.

Simultaneous Nitrification Denitrification (SND)

1. Change the scale of the nitrate plot from 24 to 48 hours. Simulate the response for 24 hours.

2. Use the main window **Project|Parameters|Kinetic…** menu command to open the **Model Parameter Editor** and change the Heterotroph and SND DO switching function values from the defaults of 0.05 mg/L to 0.5 mg/L. Continue the simulation until 48 hours, and view the difference in nitrate response.

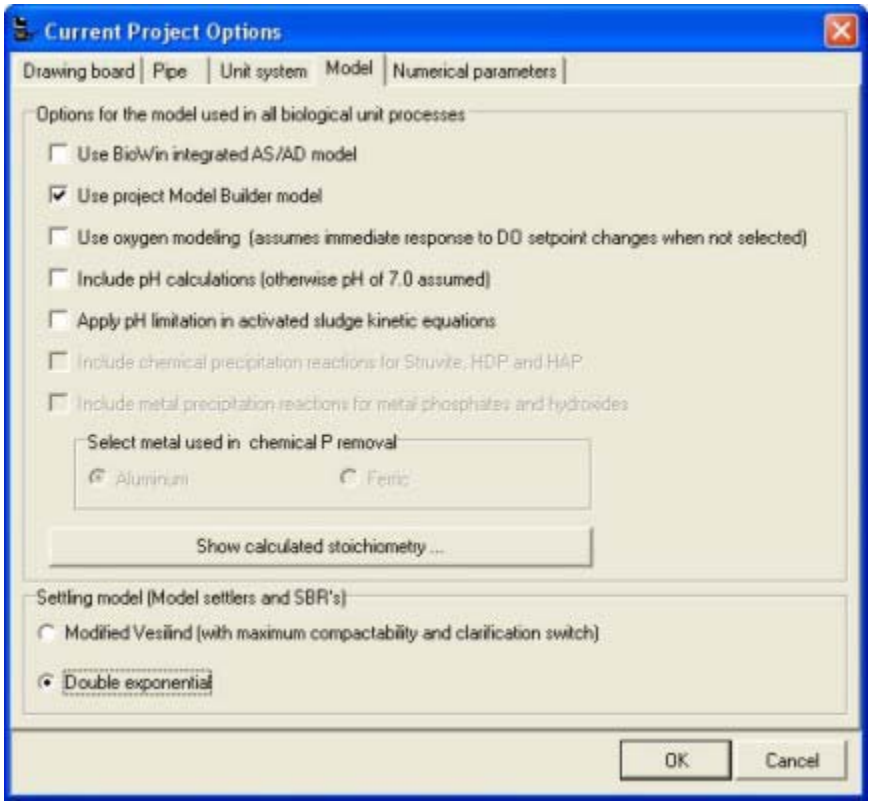
Using Builder Models

BioWin 3.0 contains the international models ASM1, ASM2d and ASM3 from the International Water Association (IWA), in addition to the general BioWin activated sludge – digestion model (ASDM),. This chapter provides guidance on how to use these models. For model details, parameter values and calibration please consult the appropriate IWA Task Group Report. BioWin also contains Add-On models, such as CaCO3 precipitation, aerobic growth of methylotrophs and slow degradation of "inert" components in very long SRT systems. After the ASM section, you will find information about these Add-On models.

General considerations for using ASMs

The ASM series have fewer state variables than the ASDM model in BioWin. The following steps must be taken in order to build a configuration using these additional models:

1. Build the configuration as usual. Take into account that the ASM series is meant to simulate only BOD removal, nitrification, denitrification, and in the case of ASM2d, biological phosphorus removal and precipitation. Do not use elements and processes that cannot be simulated with these models (digester, chemical addition, etc.)
2. In **Model Options** (accessible by clicking the button at the bottom left-hand side of the BioWin main simulator window, or from **Project|Current Project Options**), disable the BioWin integrated model, pH calculation and pH limitation. Select the **Use project Model Builder** model option (see below). DO modeling is optional.



3. In the **Project|Model Builder** dialog box, click the **Select from model library** button and select the model you wish to use. Select OK for **Overwrite Current model dialog box**….
4. In all influent elements, make sure that there are no influent concentrations (fractions) that cannot be handled by the specific model. The following table aids with identifying the various state variables that have different names in the three ASM models and their BioWin equivalent.

Description	Units	BioWin 2.1	ASM1	ASM2d	ASM3
Non-polyP heterotrophs	mgCOD/L	Z_{BH}	X_{BH}	X_H	X_{BH}
Readily biodegradable complex COD (non-VFA)	mgCOD/L	S_{BSC}	S_S	S_F	S_S
Acetate	mgCOD/L	S_{BSA}		S_A	
Soluble inert COD	mgCOD/L	S_{US}	S_I	S_I	S_I
Slowly biodegradable particulate	mgCOD/L	X_{SP}	X_S	X_S	X_S

COD					
Particulate inert COD	mgCOD/L	X_I	X_I	X_I	X_I
Endogenous products	mgCOD/L	Z_E	X_P		
Dissolved oxygen	mgO_2/L	DO	S_O	S_{O2}	S_O
Autotrophs	mgCOD/L	Z_{BA}	X_{BA}	X_{AUT}	X_{BA}
Particulate biodegradable organic nitrogen	mgN/L	X_{ON}	X_{ND}	fraction	fraction
Soluble biodegradable organic nitrogen	mgN/L	N_{OS}	S_{ND}	fraction	fraction
Ammonia N	mgN/L	NH_3-N	S_{NH}	S_{NH4}	S_{NH}
Nitrate N	mgN/L	NO_3-N	S_{NO}	S_{NO3}	S_{NO}
PolyP heterotrophs	mgCOD/L	Z_{BP}		X_{PAO}	
Stored PHA	mgCOD/L	S_{PHB}		X_{PHA}	X_{STO}
PO_4 (including Metal complexed)	mgP/L	PO_4-P		S_{PO4}	
Releasable stored polyP	mgP/L	PP_{-LO}		X_{PP}	
Hydroxy-dicalcium-phosphate	mgTSS/L	X_{HDP}		X_{MEOH}	
Hydroxy-apatite	mgTSS/L	X_{HAP}		X_{MEP}	

5. Uncheck pH calculations since the ASM series does not provide information about pH.
6. Model parameter values for Builder models are changed in the **Project | Model Builder** window.
7. This procedure replaces the BioWin model for all the elements in the configuration. It is possible to use any of the models in specific tanks, if the Model Builder unit is placed in a configuration. This way it is possible to compare different models within the same configuration.
8. Editing the model. On clicking on an entry field in the Model Builder window, an equation editor window

appears. The window will contain the equation or value if a stoichiometry matrix element was clicked. The equation or element value can be changed in this window. The “set position” tab at the bottom of the equation editor window puts the cursor to the desired position but any part of an equation can be changed by moving to the desired position using the arrows or mouse, and editing. When finished editing, click on the “Close and update” tab at the bottom right hand side of the equation editor window. Or simply close the window if no changes are desired. The Stoichiometry window displays the Petersen matrix for the model. It apparently can be edited directly- I did not explore this. Also the matrix can be edited by clicking on a value.

9. To change a rate constant value: Double click on the rate constant. The rate constant name will appear in the Name box in the middle of the pane and its value will appear in the value box below the Name box. Type the new value for the parameter. Then click on the “Add to rate constants” tab to register the change.
10. If a stoichiometric constant is to be changed, double click on it and follow the procedure in the previous point except click on the “Add to stoichiometric constants” tab to register the change.
11. The verify equations tab should be pressed after all changes to ensure that the model is verified by BioWin.
12. Be aware that combined variables (TSS, BOD, etc.) are always calculated according to the BioWin method. It is recommended that only the model state variables are used to evaluate the model’s performance. The Explorer (CTRL-E) is a useful tool to verify that only those state variables have values that are included in the model selected. Any influent concentration, state variable that is not handled by the model, will be treated as inert and will flow through the plant model unchanged.

Using Add-on Models

The BioWin Model Builder provides the user with the ability to build a new model using existing state variables (including the four user defined variables), or enhance/modify/add to the BioWin ASDM model. Three Add-on models are provided with BioWin 3.0 as examples: CaCO3 precipitation, Inert conversion and Aerobic Methylotroph growth.

87. Build the configuration as usual.
88. In Model Options (accessible by clicking the button at the bottom left-hand side of the BioWin main simulator window, or from Project|Current Project Options), select the Use project Model Builder model option. Leave the BioWin Integrated AS/AD model option and any other options selected that are desired.
89. In the Project|Model Builder dialog box, click the Select from model library button and select the model add-on you wish to use. Select OK for Overwrite Current model dialog box….
    - CaCO3 precip – Add-on to model calcium carbonate precipitation
    - Aerobic MEOH – Add-on to model aerobic growth of methylotrophs on methanol
    - Inert conversion – Adding the possibility of slow conversion of inerts to biodegradable or soluble components

Calcium carbonate precipitation Add-on

This add-on inserts a CaCO3 precipitation/dissolution kinetic reaction into the BioWin model. UD4 is precipitated calcium carbonate in mgTSS/L units. The model is based on equilibrium water chemistry involving Ca2+ and CO32- ions and a measured solubility constant for CaCO3. The extension can be used to estimate if the conditions are conducive to calcium carbonate precipitation.

Aerobic growth of methylotrophs Add-on

In the default AS/AD model methylotrophs grow exclusively under anoxic conditions. This leads to high anoxic zone volume requirements. There is indication in the literature that aerobic growth of methylotrophs is possible or even likely. This Add-on allows aerobic growth of methylotrophs, in lack of information with the same yield and growth rates as under anoxic conditions. The constants can be modified.

Inert conversion Add-on

This model add-on will extend the validity of the BioWin model for very long sludge ages or systems where alternating environmental conditions cause “inert” components to degrade. Under conditions where cumulative SRT is longer than 30 days, organic components that are normally considered inert (Xi and Ze) may start degrading. This add-on will introduce a first order reaction rate to inert organics (Xi) and endogenous residue (Ze) and convert them to slowly degradable particulates (Xsp).

The rate constant Kdxi (default value of 0.0) can be found and edited (Enter constant name and value) in the Model Builder Editor (**Project|Model Builder**).

The overall result from using this add-on is that higher VSS destruction rates can be achieved in agreement with measurements in aerobic digesters and other high sludge age systems.

Influent Inorganic Suspended Solids may dissolve under long SRT conditions. This is also added as a third rate, generating cations and anions. At the moment there are no reliable measurements to estimate a rate constant, so the default value of zero is provided.

Other Systems

For more example systems, please visit the EnviroSim web site (www.envirosim.com) where you can find many documented examples available for download.

Process Model Formulation

Process Models Used in Biowin

BioWin uses a general Activated Sludge/Anaerobic Digestion (ASDM) model which is referred to as the BioWin General Model. The BioWin General Model has fifty state variables and sixty process expressions. These expressions are used to describe the biological processes occurring in activated sludge and anaerobic digestion systems, several chemical precipitation reactions, and the gas-liquid mass transfer behavior for six gases. The model formulation requires pH determination which is described in the pH chapter. This complete model approach frees the user from having to map one model's output to another model's input which significantly reduces the complexity of building full plant models, particularly those incorporating many different process units.

This chapter provides an overview of the model and the parameters included in the model.. In some cases you will be referred to other documents that describe sections of the model in greater detail. In this section the model parameters (kinetic, stoichiometric, settling and chemical constants) are listed in full with a brief description of the model process. Parameters that are of special importance (those having a direct impact on a measurable wastewater, effluent or plant characteristics) are highlighted in the tables.

The default BioWin General Model can be augmented with additional processes or entirely replaced by other models which have been defined in the Model Builder. BioWin includes a library of Model Builder models. Details on the Model Builder are in the **Model Builder** section of the "*Common Dialog*" chapter.

For a list of the models included with BioWin, please see the **Model Library for the Model Builder** section later in this chapter.

Activated Sludge Model

The activated sludge model in BioWin contains the following functional categories.

Growth and Decay of Ordinary Heterotrophic Oganisms

Growth and Decay of Methylotrophs

Hydrolysis, Adsorption, Ammonification and Assimilative denitrification

Growth and Decay of Ammonia Oxidizing Biomass

Growth and Decay of Nitrite Oxidizing Biomass

Growth and Decay of ANaerobic AMMonia OXidizers (ANAMMOX)

Note: In an activated sludge system, under anaerobic conditions, the anaerobic processes described in the **Anaerobic digestion section** may also have a significant impact.

These modules are described in detail below.

Growth and Decay of Ordinary Heterotrophic Oganisms

Number of Processes: 11

Engineering Objective: BOD removal, denitrification

Implementation: permanent, always active in the BioWin model

Module Description:

This group of processes describes the growth of ordinary heterotrophic organisms under aerobic and anoxic conditions and the decay of these organisms under all conditions. The activated sludge model allows for direct ordinary heterotrophic aerobic growth on acetate, propionate, readily biodegradeable complex substrate and methanol. The organisms will tend to use the substrates in the order specified. Under anoxic conditions, the ordinary heterotrophs can use only three of the substrates, namely; acetate, propionate and readily biodegradable complex substrate. In the BioWin model, anoxic use of methanol is restricted to a specialized group of organisms (see anoxic methylotrophs).

The base rate expression for each of the 10 growth processes is the product of the maximum specific growth rate, the heterotrophic biomass concentration and a Monod expression for one of the substrates. This base rate is modified to account for environmental conditions (dissolved oxygen, nitrate and nitrite), nutrient limitations (ammonia, phosphate, other cations and anions), pH inhibition and substrate preference weighting. BioWin uses ammonia as a nitrogen source for cell synthesis with all of the substrates under aerobic, anoxic and anaerobic conditions. At low ammonia concentrations BioWin allows for assimilative ammonia production from either nitrate or nitrite in order to satisfy synthesis demands.

Under anoxic conditions the base rate is also multiplied by the anoxic growth factor and preferentially uses nitrite as the electron acceptor. The decay process has a rate that varies according to the electron acceptor environment.

Model parameters affecting the performance of this module are listed below:

Kinetic Parameters

Menu Location: **Project|Parameters|Kinetic|OHOs**

Name	Default Value	Unit	Explanation
Max. spec. growth rate	3.2	d^{-1}	Determines the maximum specific growth rate of heterotrophs. Substrate and nutrient limitations will decrease the growth rate. This parameter is sensitive only in very

			high loaded plants (short SRT), and determines maximum BOD removal capacity.
Substrate half sat.	5.0	mgCOD/L	This parameter impacts the residual soluble substrate concentration in the effluent. The value is usually low in normal municipal plants.
Anoxic growth factor	0.5	-	This parameter decreases the maximum specific growth rate under anoxic conditions. Substrate and nutrient limitations may further reduce the growth rate.
Aerobic decay	0.62	d^{-1}	Decay rate constant under aerobic conditions. This parameter impacts the endogenous respiration rate and VSS destruction during aerobic stabilization.
Anoxic/anaerobic decay	0.3	d^{-1}	Decay rate constant in the absence of oxygen.

Stoichiometric Parameters

Menu Location: **Project|Parameters|Stoichiometric| OHOs**

Name	Default Value	Unit	Explanation
Yield (Aerobic)	0.666	mgCOD/mgCOD	Amount of biomass COD produced using one unit of readily biodegradable complex substrate COD. The remaining COD is oxidized. This parameter is very stable in municipal plants and seldom needs adjustment. In case there is a mismatch between measured and simulated sludge production and OUR, try adjusting the influent fup (unbiodegradable particulate COD fraction) parameter or check wastage and SRT.
N in Biomass	0.07	mgN/mgCOD	N content of heterotrophs. This parameter impacts the nitrogen available for nitrification and therefore oxygen demand.
N in Inert	0.07	mgN/mgCOD	N content of endogenous residue from heterotrophic decay.
P in Biomass	0.022	mgP/mgCOD	P content of heterotrophs. This parameter influence the P removal in non bio-P systems, and the P content of the sludge.
P in Inert	0.022	mgP/mgCOD	P content of endogenous residue from heterotrophic decay.

Endogenous Residue	0.08	-	Fraction of biomass that becomes inert upon decay.
COD:VSS Ratio	1.42	mgCOD/mgVSS	Conversion factor between biomass as measured in COD and its VSS content. This value is relatively stable for biomass.
Yield (anoxic)	0.54	mgCOD/mgCOD	Biomass yield on readily biodegradable complex substrate COD under anoxic conditions.
Yield propionic (aerobic)	0.5	mgCOD/mgCOD	Biomass yield on propionic acid COD under aerobic conditions.
Yield propionic (anoxic)	0.41	mgCOD/mgCOD	Biomass yield on propionic acid COD under anoxic conditions.
Yield acetic (aerobic)	0.4	mgCOD/mgCOD	Biomass yield on acetic acid COD under aerobic conditions.
Yield acetic (anoxic)	0.32	mgCOD/mgCOD	Biomass yield on acetic acid COD under anoxic conditions.
Yield methanol (Aerobic)	0.5	mgCOD/mgCOD	Biomass yield on methanol COD under anoxic conditions.

pH Inhibition

Menu Location: **Project|Parameters|Kinetic|pH**

Name	Default Value	Unit	Explanation
Heterotrophs low pH limit	4.0	pH units	At a pH equal to this value the growth rate of ordinary heterotrophic biomass will be reduced by 50%. Heterotrophs exhibit tolerance for pH changes – hence the wide pH range.
Heterotrophs high pH limit	10.0	pH units	At a pH equal to this value the growth rate of ordinary heterotrophic biomass will be reduced by 50%.

Switching Functions

Menu Location: **Project|Parameters|Kinetic|Switches**

Name	Default Value	Unit	Explanation
Heterotrophic DO half sat.	0.05	mgO_2/L	This constant is used to switch off aerobic OHO activity under low DO conditions (that is in anaerobic and anoxic reactors).
Aerobic denit. DO half sat.	0.05	mgO_2/L	This constant is used to turn anoxic OHO growth processes on under low dissolved oxygen conditions. [Simultaneous or aerobic denitrification switch.]
Anoxic NO3	0.1	mgN/L	This constant is used to switch off anoxic growth processes using nitrate under low

half sat.			nitrate conditions.
Anoxic NO2 half sat.	0.1	mgN/L	This constant is used to switch off anoxic growth processes using nitrite under low nitrite conditions.
NH3 nutrient half sat.	0.001	mgN/L	This constant is used to slow all biomass growth processes at low ammonia-N concentrations (N nutrient limiting conditions – see assimilative denitrification).
P nutrient half sat.	0.001	mgP/L	This constant is used to slow the growth of biomass when there is no phosphorus available as nutrient.
Synthesis anion/cation half sat.	0.01	meq/L	Half saturation concentration for anions and cations.

Growth and Decay of Methylotrophs

Number of Processes: 3

Engineering Objective: denitrification using methanol

Implementation: permanent, always active in the BioWin model

Module Description:

These processes describe the growth and decay of specialized heterotrophs using methanol under anoxic conditions. In the BioWin model anoxic methylotrophs can only growth under anoxic conditions using methanol as substrate and either nitrate or nitrite as an electron acceptor. They require a minimum "anoxic SRT " (similar in concept to the minimum aerobic SRT required by autotrophs) to maintain themselves in the activated sludge system without washing out. Nitrogen source for cell synthesis of these microorganisms is ammonia. The base rate expression for this growth process is the product of the maximum specific growth rate, the anoxic methylotrophs concentration and a Monod expression for methanol. This base rate is modified to account for environmental conditions (dissolved oxygen, nitrate and nitrite), nutrient limitations (ammonia, phosphate, other cations and anions), pH inhibition and electron acceptor preference weighting. BioWin preferentially uses nitrite as an electron acceptor.

The single decay rate varies between an aerobic value and an unaerated value depending oxygen concentration.

Model parameters are listed in:

Kinetic Parameters

Menu Location: **Project|Parameters|Kinetic|Methylotrophs**

Name	Default Value	Unit	Explanation
Max. spec. growth rate of methanol	1.3	d^{-1}	Determines the maximum specific growth rate of methylotrophs.

utilizers			Substrate and nutrient limitations will decrease the growth rate. This parameter will determine the necessary anoxic SRT to maintain a viable denitrifying population on methanol.
Methanol half sat.	0.5	mgCOD/L	This parameter impacts the residual methanol concentration bleeding out of the anoxic tank. The value is usually very low in normal municipal plants (once the suitable population has been established).
Aerobic decay rate of methanol utilizers	0.04	d^{-1}	Decay rate constant under aerobic conditions. This parameter impacts the minimum anoxic SRT.
Anoxic/anaerobic decay rate of methanol utilizers	0.03	d^{-1}	Decay rate constant in the absence of oxygen. This parameter impacts the minimum anoxic SRT.

Stoichiometric Parameters

Menu Location: **Project|Parameters|Stoichiometric|Methylotrophs**

Name	Default Value	Unit	Explanation
Yield (anoxic)	0.4	mgCOD/mgCOD	Anoxic methylotrophic biomass yield on readily methanol COD under anoxic conditions. This parameter has significant impact on the methanol dosage required to denitrify 1 mgN nitrate (3.2mg methanol/mgNO_3-N by default)
N in Biomass	0.07	mgN/ mgCOD	N content of anoxic methylotrophs.
N in Inert	0.07	mgN/ mgCOD	N content of endogenous residue from anoxic methylotrophic organism decay.
P in Biomass	0.022	mgP/ mgCOD	P content of anoxic methylotrophs.
P in Inert	0.022	mgP/ mgCOD	P content of endogenous residue from anoxic methylotrophic organism decay.
Endogenous Residue	0.08	-	Fraction of biomass that becomes inert upon decay.
COD:VSS Ratio	1.42	mgCOD/mgVSS	Conversion factor between biomass as measured in COD and its VSS content. This value is relatively stable for biomass.

pH Inhibition

Menu Location: **Project|Parameters|Kinetic|pH**

Name	Default Value	Unit	Explanation
√Methanol tilizers low pH limit	4.0	pH units	At a pH equal to this value the growth rate of methylotrophs will be reduced by 50%.
Methanol utilizers high pH limit	10.0	pH units	At a pH equal to this value the growth rate of methylotrophs will be reduced by 50%.

Switching Functions

Menu Location: **Project|Parameters|Kinetic|Switches**

Name	Default Value	Unit	Explanation
Heterotrophic DO half sat.	0.05	mgO_2/L	This constant is used to switch off aerobic activity under low DO conditions (that is in anaerobic and anoxic reactors).
Anoxic NO3 half sat.	0.1	mgN/L	This constant is used to switch off anoxic growth processes using nitrate under low nitrate conditions.
Anoxic NO2 half sat.	0.1	mgN/L	This constant is used to switch off anoxic growth processes using nitrite under low nitrite conditions.
NH3 nutrient half sat.	0.001	mgN/L	This constant is used to slow all biomass growth processes at low ammonia-N concentrations (N nutrient limiting conditions – see assimilative denitrification).
P nutrient half sat.	0.001	mgP/L	This constant is used to slow the growth of biomass when there is no phosphorus available as nutrient.
Synthesis anion/cation half sat.	0.01	meq/L	Half saturation concentration for anions and cations.

Hydrolysis, Adsorption, Ammonification and Assimilative denitrification

Number of Processes: 7

Engineering Objective: Conversion of organic, nitrogen and phosphorus fractions

Implementation: permanent, always active in the BioWin model

Module Description:

These processes are discussed here separately from the organism groupings because they involve more that one organism type (in general both the ordinary heterotrophic organisms and the phosphate accumulating organisms.

Hydrolysis of biodegradable particulate organic substrate to readily biodegradable complex substrate: The base rate is the product of the hydrolysis rate constant, the sum of the ordinary heterotrophs and the phosphate accumulating organisms, and a Monod expression for ratio of particulate substrate to organism COD. There is an efficiency factor applied for anoxic conditions and another for anaerobic conditions.

Hydrolysis of biodegradable particulate organic nitrogen and phosphorus: The hydrolysis of biodegradable particulate nitrogen (phosphorus) is assumed to proceed at the same rate as the biodegradable particulate organics but is adjusted by the ratio of biodegradable particulate organic nitrogen (phosphorus) to biodegradable particulate organic.

Adsorption or flocculation of colloidal organic material to particulate organic material (occurring spontaneously as opposed to chemically facilitated flocculation with metal (ferric or alum) addition: The rate is the product of the adsorption rate constant, the colloidal substrate concentration and the sum of the ordinary heterotrophs and the phosphate accumulating organism concentrations. The rate is decreased as the ratio of particulate substrate to organism COD approaches the maximum adsorption ratio constant.

Ammonification of soluble organic nitrogen to ammonia: The rate is the product of the ammonification rate constant, the soluble organic nitrogen concentration and the sum of the ordinary heterotrophs and the phosphate accumulating organism concentrations.

Assimilative denitrification of nitrate or nitrite to ammonia for synthesis: BioWin allows for the production of ammonia for synthesis by any organisms under low ammonia conditions (as ammonia becomes limiting for growth). The assimilative process will use nitrite if it is available otherwise it will use nitrate. The base rate is the product of the assimilation rate constant and the total organism COD. This base rate is modified to account for environmental conditions (off with ammonia, and selecting between nitrate and nitrite).

Model parameters are listed in:

Kinetic Parameters

Menu Location: **Project|Parameters|Kinetic|OHOs**

Name	Default Value	Unit	Explanation
Hydrolysis rate (AS)	2.1	d^{-1}	Rate constant for hydrolysis of slowly degradable organics into readily degradable substrate; for activated sludge.
Hydrolysis half sat. (AS)	0.06	-	Monod half saturation constant for the regulation of hydrolysis rate, expressed in terms of particulate substrate to heterotrophic biomass ratio; for activated sludge.
Anoxic hydrolysis factor	0.28	-	Rate reduction factor for hydrolysis under anoxic conditions.
Anaerobic hydrolysis factor	0.5	-	Rate reduction factor for hydrolysis under anaerobic conditions in activated sludge.
Adsorption rate of colloids	0.8	d^{-1}	Conversion rate of colloidal material to particulate.

Ammonification rate	0.04	d^{-1}	Conversion rate of soluble organic nitrogen compounds to ammonia
Assimilative nitrate/nitrite reduction rate	0.5	d^{-1}	Conversion rate of nitrite and/or nitrate to ammonia under ammonia limited conditions
Hydrolysis rate (AD)	0.1	d^{-1}	Hydrolysis rate of particulate organics in anaerobic digesters
Hydrolysis half sat. (AD)	0.15	-	Monod half saturation constant for regulation of hydrolysis rate in anaerobic digesters, expressed in terms of particulate substrate to heterotrophic biomass ratio

Stoichiometric Parameters

None

Switching Functions

Menu Location: **Project|Parameters|Kinetic|Switches**

Name	Default Value	Unit	Explanation
Heterotrophic DO half sat.	0.05	mgO_2/L	This constant is used to switch off aerobic activity under low DO conditions (that is in anaerobic and anoxic reactors).
Anoxic NO3 half sat.	0.1	mgN/L	This constant is used to switch off anoxic growth processes using nitrate under low nitrate conditions.
Anoxic NO2 half sat.	0.1	mgN/L	This constant is used to switch off anoxic growth processes using nitrite under low nitrite conditions.
NH3 nutrient half sat.	0.001	mgN/L	This constant is used to slow all biomass growth processes at low ammonia-N concentrations (N nutrient limiting conditions – see assimilative denitrification**).**

Growth and Decay of Ammonia Oxidizing Biomass (AOB)

Number of Processes: 2

Engineering Objective: Nitrification

Implementation: permanent, always active in the BioWin model

Module Description:

This biomass grows by oxidizing ammonia to nitrite and using the energy to synthesis organic material from inorganic carbon (fixing CO_2). Nitrogen source for cell synthesis is ammonia.

The base rate expression for the growth process is the product of the maximum specific growth rate, the ammonia oxidizing biomass concentration and a Monod expression for ammonia. This base rate is modified to account for environmental conditions (off at low dissolved oxygen and inhibited by nitrous acid), nutrient limitations (phosphate, inorganic carbon, other cations and anions) and pH inhibition.

The decay rate varies between an aerobic value and an anoxic/anaerobic value depending dissolved oxygen concentrations.

Model parameters are listed in:

Kinetic Parameters

Menu Location: **Project|Parameters|Kinetic|AOB**

Name	Default Value	Unit	Explanation
Max. spec. growth rate	0.9	d-1	Determines the maximum specific growth rate of ammonia oxidizing biomass. Substrate and nutrient limitations will decrease the growth rate. This parameter has a direct impact on the nitrification capacity.
Substrate (NH_4) half sat.	0.7	mgN/L	This parameter impacts the residual ammonia concentration in the effluent. The value is usually low in normal municipal plants.
Aerobic decay rate	0.17	d-1	Decay rate constant under aerobic conditions for ammonia oxidizing biomass.
Anoxic/anaerobic decay rate	0.08	d-1	Decay rate constant under non-aerobic conditions for ammonia oxidizing biomass.
$KiHNO_2$	0.005	mmol/L	Nitrous acid inhibition concentration.

Stoichiometric Parameters

Menu Location: **Project|Parameters|Stoichiometric|AOB**

Name	Default Value	Unit	Explanation
Yield	0.15	mgCOD/mgN	AOB COD produced by oxidizing 1 mg of ammonia.
N in biomass	0.07	mgN/ mgCOD	N content of AOB.
N in inert	0.07	mgN/ mgCOD	N content of endogenous residue from AOB decay.
P in biomass	0.022	mgP/ mgCOD	P content of AOB.
P in inert	0.022	mgP/ mgCOD	P content of endogenous residue from AOB decay.
Fraction to endogenous	0.08	-	Fraction of biomass that becomes inert upon decay.

residue			
COD:VSS ratio	1.42	mgCOD/mgVSS	Conversion factor between biomass as measured in COD and its VSS content. This value is relatively stable for biomass.

pH Inhibition

Menu Location: **Project|Parameters|Kinetic|pH**

Name	Default Value	Unit	Explanation
Autotrophs low pH limit	5.5	pH units	At a pH equal to this value the growth rate of AOB, NOB and ANAMMOX will be reduced by 50%.
Autotrophs high pH limit	9.5	pH units	At a pH equal to this value the growth rate of AOB, NOB and ANAMMOX will be reduced by 50%.

Switching Functions

Menu Location: **Project|Parameters|Kinetic|Switches**

Name	Default Value	Unit	Explanation
Ammonia oxidizer DO half sat.	0.25	mgO_2/L	This parameter is used to switch off ammonia oxidation by AOB under low DO conditions.
P nutrient half sat.	0.001	mgP/L	This parameter is used to switch off the growth of biomass when there is no phosphorus available as nutrient.
Autotroph CO2 half sat.	0.1	mmol/L	This parameter is used to switch off the growth of AOB, NOB and ANAMMOX when there is little inorganic carbon available.
Synthesis anion/cation half sat.	0.01	meq/L	Half saturation concentration for anions and cations.

Growth and Decay of Nitrite Oxidizing Biomass (NOB)

Number of Processes: 2

Engineering Objective: Nitrification

Implementation: permanent, always active in the BioWin model

Module Description:

This biomass grows by oxidizing nitrite to nitrate and using the energy to synthesis organic material from inorganic carbon (fixing CO_2). Nitrogen source for cell synthesis is ammonia.

The base rate expression for the growth process is the product of the maximum specific growth rate, the nitrite oxidizing biomass concentration and a Monod expression for nitrite. This base rate is modified to account for environmental conditions (off at low dissolved oxygen and inhibited by ammonia), nutrient limitations (ammonia, phosphate, inorganic carbon, other cations and anions) and pH inhibition.

The decay rate varies between an aerobic value and an anoxic/anaerobic value depending dissolved oxygen concentrations.

Model parameters are listed in:

Kinetic Parameters

Menu Location: **Project|Parameters|Kinetic|NOB**

Name	Default Value	Unit	Explanation
Max. spec. growth rate	0.7	d-1	Determines the maximum specific growth rate of nitrite oxidizing biomass. Substrate, nutrient limitations and environmental conditions will decrease the growth rate. This parameter has a direct impact on the nitrification capacity.
Substrate (NO_2) half sat.	0.05	mgN/L	This parameter impacts the residual nitrite concentration in the effluent. The value is usually very low in normal municipal plants.
Aerobic decay rate	0.17	d-1	Decay rate constant under aerobic conditions for nitrite oxidizing biomass.
Anoxic/anaerobic decay rate	0.08	d-1	Decay rate constant under non-aerobic conditions for nitrite oxidizing biomass.
$KiHNH_3$	0.075	mmol/L	NH3 inhibition concentration.

Stoichiometric Parameters

Menu Location: **Project|Parameters|Stoichiometric|NOB**

Name	Default Value	Unit	Explanation
Yield	0.09	mgCOD/mgN	NOB COD produced by oxidizing 1 mg of nitrite N.
N in biomass	0.07	mgN/ mgCOD	N content of NOB.
N in inert	0.07	mgN/ mgCOD	N content of endogenous residue from NOB decay.
P in biomass	0.022	mgP/ mgCOD	P content of NOB.
P in inert	0.022	mgP/ mgCOD	P content of endogenous residue from NOB decay.

Fraction to endogenous residue	0.08	-	Fraction of biomass that becomes inert upon decay.
COD:VSS ratio	1.42	mgCOD/mgVSS	Conversion factor between biomass as measured in COD and its VSS content. This value is relatively stable for biomass.

pH Inhibition

Menu Location: **Project|Parameters|Kinetic|pH**

Name	Default Value	Unit	Explanation
Autotrophs low pH limit	5.5	pH units	At a pH equal to this value the growth rate of AOB, NOB and ANAMMOX will be reduced by 50%.
Autotrophs high pH limit	9.5	pH units	At a pH equal to this value the growth rate of AOB, NOB and ANAMMOX will be reduced by 50%.

Switching Functions

Menu Location: **Project|Parameters|Kinetic|Switches**

Name	Default Value	Unit	Explanation
Nitrite oxidizer DO half sat.	0.5	mgO_2/L	This parameter is used to switch off nitrite oxidation by NOB under low DO conditions.
P nutrient half sat.	0.001	mgP/L	This parameter is used to switch off the growth of biomass when there is no phosphorus available as nutrient.
Autotroph CO2 half sat.	0.1	mmol/L	This parameter is used to switch off the growth of AOB, NOB and ANAMMOX when there is little inorganic carbon available.
Synthesis anion/cation half sat.	0.01	meq/L	Half saturation concentration for anions and cations.

Growth and Decay of ANaerobic AMMonia OXidizers (ANAMMOX)

Number of Processes: 2

Engineering Objective: Nitrification

Implementation: permanent, always active in the BioWin model

Module Description:

This biomass grows by converting ammonia and nitrite to nitrogen gas and nitrate. The energy from this process is used to synthesis organic material from inorganic carbon (fixing CO_2). Nitrogen source for cell synthesis is ammonia.

The base rate expression for the growth process is the product of the maximum specific growth rate, the ANAMMOX concentration, a Monod expression for ammonia and a Monod expression for nitrite. This base rate is modified to account for environmental conditions (switched off under aerobic conditions and inhibited by nitrite), nutrient limitations (phosphate, inorganic carbon, other cations and anions) and pH inhibition.

The decay rate varies between an aerobic and anoxic/anaerobic value depending dissolved oxygen concentrations. Nitrite toxicity is modeled by increasing the decay rate by the product of the nitrite sensitivity constant and the nitrite concentration.

Model parameters are listed in:

Kinetic Parameters

Menu Location: **Project|Parameters|Kinetic|ANAMMOX**

Name	Default Value	Unit	Explanation
Max. spec. growth rate	0.1	d-1	Determines the maximum specific growth rate of ANAMMOX. Substrate, nutrient limitations and environmental conditions will decrease the growth rate. This parameter has a direct impact on the nitrification capacity.
Substrate (NH_4) half sat.	2.0	mgN/L	Ammonia half saturation concentration for ANNAMOX.
Substrate (NO_2) half sat.	1.0	mgN/L	Nitrite half saturation concentration for ANNAMOX.
Aerobic decay rate	0.019	d-1	Decay rate constant under aerobic conditions for ANNAMOX.
Anoxic/anaerobic decay rate	0.0095	d-1	Decay rate constant under non-aerobic conditions for ANNAMOX.
Ki Nitrite	1000.0	mgN/L	Nitrite inhibition concentration.
Nitrite sensitivity constant	0.016	L/ (d mgN)	Nitrite toxicity constant.

Stoichiometric Parameters

Menu Location: **Project|Parameters|Stoichiometric|ANNAMOX**

Name	Default Value	Unit	Explanation
Yield	0.114	mgCOD/mgN	ANNAMOX produced by oxidizing 1 mg of ammonia.
Nitrate production	2.28	mgN/mgCOD	Nitrate production yield
N in biomass	0.07	mgN/ mgCOD	N content of ANNAMOX.

N in inert	0.07	mgN/ mgCOD	N content of endogenous residue from ANNAMOX decay.
P in biomass	0.022	mgP/ mgCOD	P content of ANNAMOX.
P in inert	0.022	mgP/ mgCOD	P content of endogenous residue from ANNAMOX decay.
Fraction to endogenous residue	0.08	-	Fraction of biomass that becomes inert upon decay.
COD:VSS ratio	1.42	mgCOD/mgVSS	Conversion factor between biomass as measured in COD and its VSS content. This value is relatively stable for biomass.

pH Inhibition

Menu Location: **Project|Parameters|Kinetic|pH**

Name	Default Value	Unit	Explanation
Autotrophs low pH limit	5.5	pH units	At a pH equal to this value the growth rate of AOB, NOB and ANAMMOX will be reduced by 50%.
Autotrophs high pH limit	9.5	pH units	At a pH equal to this value the growth rate of AOB, NOB and ANAMMOX will be reduced by 50%.

Switching Functions

Menu Location: **Project|Parameters|Kinetic|Switches**

Name	Default Value	Unit	Explanation
Anaerobic ammonia oxidizer DO half sat.	0.01	mgO_2/L	This parameter is used to switch on anaerobic ammonia oxidation by ANNAMOX under very low DO conditions.
P nutrient half sat.	0.001	mgP/L	This parameter is used to switch off the growth of biomass when there is no phosphorus available as nutrient.
Autotroph CO2 half sat.	0.1	mmol/L	This parameter is used to switch off the growth of AOB, NOB and ANAMMOX when there is little inorganic carbon available.
Synthesis anion/cation half sat.	0.01	meq/L	Half saturation concentration for anions and cations.

Growth and Decay of Phosphorus Accumulating Organisms

Number of Processes: 15

Engineering Objective: Biological phosphorus removal

Implementation: permanent, always active in the BioWin model

Module Description:

This group of processes describes the growth and decay of polyphosphate accumulating organisms (PAOs) under all conditions. This includes descriptions of aerobic and anoxic growth, VFA sequestration and polyphosphate lysis.

There are two maximum specific growth rates for PAOs under aerobic conditions. The lower growth rate constant is used under P limited conditions and has a different stoichiometry (no polyphosphate storage). There are also two anoxic growth processes, one uses nitrate and the other nitrite. Growth processes under phosphate rich conditions result in uptake of phosphate, and balancing calcium magnesium ions and other cations. A lack of these ions will stop the growth processes by appropriate Monod switches. For all of these growth processes, the base growth rate is the product of the maximum specific rate constant, the PAO concentration and a Monod on the ratio PAO COD to PHA COD. This base rate is modified to account for environmental conditions (dissolved oxygen, nitrate and nitrite), nutrient limitations (ammonia, anions, cations, for polyphosphate storage magnesium, and calcium are also required) and pH inhibition. BioWin uses ammonia as a nitrogen source for cell synthesis under aerobic, anoxic and anaerobic conditions. At low ammonia concentrations BioWin allows for assimilative ammonia production from either nitrate or nitrite in order to satisfy synthesis demands.

Under anoxic conditions the base rate is also multiplied by an anoxic growth factor and preferentially uses nitrite as the electron acceptor.

The PAOs use energy store in polyphosphate to sequester acids and store them as store PHA. In the BioWin model the PAOs can use both acetate and propionate for this process. The base sequestration rate is the product of the sequestration rate constant, the PAO concentration and a Monod on the appropriate substrate (acetate or propionate) The base rate is switched on the availability of the stored polyphosphate (ratio of low molecular weight polyphosphate to PAO COD).

There are two decay processes (aerobic and anoxic/anaerobic). Associated with each decay process is a lysis process for PHA, low and high molecular weight polyphosphate. The lysis rates are directly proportional to the decay rate itself.

There is one Poly-P cleavage process for anaerobic maintenance that releases phosphate if no oxygen is present. This process is proportional to the anoxic/anaerobic decay rate.

Model parameters are listed in:

Kinetic Parameters

Menu Location: **Project|Parameters|Kinetic|PAOs**

Name	Default Value	Unit	Explanation
Max. spec. growth rate	0.95	d^{-1}	Determines the maximum attainable growth rate of phosphorus accumulating heterotrophic

			organisms if no substrate, DO or P limitation occurs.
Max. spec. growth rate, P-limited	0.42	d^{-1}	Determines the maximum attainable growth rate of phosphorus accumulating heterotrophic organisms under phosphorus limiting conditions.
Substrate half sat.	0.1	mgCOD/L	Half saturation constant for PHA, used as substrate by phosphorus accumulating organisms.
Substrate half sat., P-limited	0.05	mgCOD/L	Half saturation constant for PHA, under phosphorus limiting conditions.
Magnesium half sat.	0.1	mgMg/L	Half saturation constant for Magnesium storage during poly-P synthesis.
Cation half sat.	0.1	meq/L	Half saturation constant for cation (primarily potassium) storage during poly-P synthesis.
Aerobic decay rate	0.1	d^{-1}	Decay rate constant in aerobic conditions.
Anaerobic decay rate	0.04	d^{-1}	Decay rate constant when there is no oxygen available.
Sequestration rate	6.0	d^{-1}	Rate constant for VFA sequestration to form PHA (stored substrate).
Anoxic growth factor	0.33	-	Together with the max. spec. growth rate, determines the maximum attainable growth rate if nitrate is available only as electron acceptor (and no substrate limitation occurs).

Stoichiometric Parameters

Menu Location: **Project|Parameters|Stoichiometric|PAOs**

Name	Default Value	Unit	Explanation
Yield (aerobic)	0.639	mgCOD/mgCOD	Amount of biomass produced using one unit of substrate under aerobic conditions. The rest of the substrate will be oxidized.
Yield (anoxic)	0.52	mgCOD/mgCOD	Amount of biomass produced using one unit of substrate under anoxic conditions.
Aerobic P/PHA uptake	0.95	mgP/mgCOD	Amount of P stored per unit of PHA oxidized in aerobic conditions
Anoxic P/PHA uptake	0.35	mgP/mgCOD	Amount of P stored per unit of PHA in anoxic conditions.
Yield of PHA on sequestration	0.889	mgCOD/mgCOD	Amount of PHA stored when 1 mg of acetate or propionate is sequestered.
N in biomass	0.07	mgN/ mgCOD	N content of phosphorus accumulating organisms. Has a significant effect on nitrogen availability for nitrification and

			therefore oxygen demand.
N in part. inert	0.07	mgN/ mgCOD	N content of endogenous residue originating from phosphorus accumulating organism decay.
N in sol. inert	0.07	mgN/ mgCOD	N content of soluble inert organics originating from phosphorus accumulating organism decay.
P in biomass	0.022	mgP/ mgCOD	P content of phosphorus accumulating organisms, not including P stored in the form of Poly-P
P in part. inert	0.022	mgP/ mgCOD	P content of endogenous residue originating from phosphorus accumulating organism decay.
Fraction to inert part.	0.25	-	Fraction of biomass that becomes particulate inert upon decay.
Fraction to inert sol.	0.2	-	Fraction of biomass that becomes soluble inert upon decay.
P/Ac release ratio	0.49	mgP/mgCOD	Amount of P released for one mg of acetate sequestered in the form of PHA
COD:VSS Ratio	1.42	mgCOD/mgVSS	Conversion factor between biomass as measured in COD and its VSS content. This value is relatively stable for biomass.
Yield of low PP	0.94	mgP/mgP	Fraction of P stored in releasable poly-P form (rest of P is stored in high molecular weight, non-releasable poly-P)

pH Inhibition

Menu Location: **Project|Parameters|Kinetic|pH**

Name	Default Value	Unit	Explanation
PolyP heterotrophs low pH limit	4.0	pH units	At a pH equal to this value the growth rate of poly phosphate accumulating biomass will be reduced by 50%.
Poly P heterotrophs high pH limit	10.0	pH units	At a pH equal to this value the growth rate of poly phosphate accumulating biomass will be reduced by 50%.

Switching Functions

Menu Location: **Project|Parameters|Kinetic|Switches**

Name	Default Value	Unit	Explanation
Heterotrophic	0.05	mgO_2/L	This constant is used to switch off aerobic

DO half sat.			OHO activity under low DO conditions (that is in anaerobic and anoxic reactors).
Anoxic NO3 half sat.	0.1	mgN/L	This constant is used to switch off anoxic growth processes using nitrate under low nitrate conditions.
Anoxic NO2 half sat.	0.1	mgN/L	This constant is used to switch off anoxic growth processes using nitrite under low nitrite conditions.
NH3 nutrient half sat.	0.001	mgN/L	This constant is used to slow all biomass growth processes at low ammonia-N concentrations (N nutrient limiting conditions – see assimilative denitrification).
PolyP half sat.	0.01	mgP/L	This constant stops sequestration of VFA and P release as the ratio of low molecular weight polyphosphate to PAO COD falls.
VFA sequestration half sat.	5.0	mgCOD/L	This is the half saturation concentration for the sequestration of acetate and propionate.
P uptake half sat.	0.15	mgP/L	This constant stops growth with poly phosphate storeage at low soluble phosphate concentrations. This constant will have an impact on the effluent soluble P concentration in a bio-P system.
P nutrient half sat.	0.001	mgP/L	This constant is used to slow the growth of biomass when there is no phosphorus available as nutrient.
Synthesis anion/cation half sat.	0.01	meq/L	Half saturation concentration for anions and cations.

Other Parameters

Menu Location: **Project|Parameters|Other|General**

Name	Default Value	Unit	Explanation
Mg to P mole ratio in polyphosphate	0.3	molMg/molP	Mole ratio of magnesium to phosphorus in stored polyphosphate. This magnesium is released when polyphosphate is used (together with the phosphate release).
Cation to P mole ratio in polyphosphate	0.3	meq/mmolP	Mole ratio of other cations (primarily potassium) to phosphorus in stored polyphosphate. These cations are released when polyphosphate is used (together with the phosphate release).
Ca to P mole ratio in polyphosphate	0.05	molCa/molP	Mole ratio of calcium to phosphorus in stored polyphosphate. This calcium is released when polyphosphate is used (together with the phosphate release).

Anaerobic Digestion Model

The anaerobic digestion model in BioWin contains the following functional categories, modules

Heterotrophic Growth through Fermentation

Growth and Decay of Propionic Acetogens

Growth and Decay of Methanogens

These modules are described in detail below.

Heterotrophic Growth through Fermentation

Number of Processes: 2

Engineering Objective: VFA generation (fermenters, digesters)

Implementation: permanent, always active in the BioWin model

Module Description:

There are two pathways for the fermentation of readily biodegradable (complex) substrate to acetate, propionate, carbon dioxide and hydrogen. The dominant pathway is governed by the dissolved hydrogen concentration. These processes are mediated by the ordinary heterotrophic organisms.

The base rate expression for the fermentation growth process is the product of the maximum specific growth rate constant, the heterotrophic biomass concentration and a Monod expression for the readily biodegradable (complex) substrate. This base rate is modified to account for nutrient limitations (ammonia, phosphate, other cations and anions) and pH inhibition. In activated sludge vessels there is an anaerobic growth factor applied. BioWin uses ammonia as a nitrogen source for cell synthesis.

Model parameters affecting the performance of this module are listed below.

Kinetic Parameters

Menu Location: **Project|Parameters|Kinetic|OHOs**

Name	Default Value	Unit	Explanation
Anoxic/anaerobic decay*	0.3	d^{-1}	Decay rate constant when there is no oxygen available.
Fermentation rate	3.2	d^{-1}	Maximum specific growth rate of heterotrophs under anaerobic conditions.
Fermentation half sat.	5.0	mgCOD/L	Half saturation of complex substrate under anaerobic conditions
Anaerobic growth factor (AS)	0.125	-	Growth rate reduction under anaerobic conditions in activated sludge

Stoichiometric Parameters

Menu Location: **Project|Parameters|Stoichiometric|OHOs**

Name	Default Value	Unit	Explanation
Yield (fermentation low H2)	0.1	mgCOD/mgCOD	Amount of biomass produced on one unit of complex substrate fermented, under low H_2 concentration.
Yield (fermentation high H2)	0.1	mgCOD/mgCOD	Amount of biomass produced on one unit of complex substrate fermented, under high H_2 concentration.
H2 yield (fermentation low H2)	0.35	mgCOD/mgCOD	Amount of hydrogen produced on one unit of complex substrate fermented, under low H_2 concentration.
H2 yield (fermentation high H2)	0.0	mgCOD/mgCOD	Amount of hydrogen produced on one unit of complex substrate fermented, under high H_2 concentration.
Propionate yield (fermentation low H2)	0.0	mgCOD/mgCOD	Amount of propionate produced on one unit of complex substrate fermented, under low H_2 concentration.
Propionate yield (fermentation high H2)	0.7	mgCOD/mgCOD	Amount of propionate produced on one unit of complex substrate fermented, under high H_2 concentration.
CO2 yield (fermentation low H2)	0.7	mmolCO_2/mmolHAC	Moles of CO_2 produced per mole of acetate formed at low dissolved H_2 concentrations.
CO2 yield (fermentation high H2)	0.0	mmolCO_2/mmolHAC	Moles of CO_2 produced per mole of acetate formed at high dissolved H_2 concentrations.
N in Biomass	0.07	mgN/ mgCOD	N content of biomass.
N in Inert	0.07	mgN/ mgCOD	N content of endogenous residue originating from heterotrophic decay.
P in Biomass	0.022	mgP/ mgCOD	P content of heterotrophs. This parameter influences the P removal in non bio-P systems, and the P content of the sludge.
P in Inert	0.022	mgP/ mgCOD	P content of endogenous residue originating from heterotrophic decay.
Endogenous Residue	0.08	-	Fraction of biomass that becomes inert upon decay.
COD:VSS Ratio	1.42	mgCOD/mgVSS	Conversion factor between biomass as measured in COD and its VSS content. This value is relatively stable for biomass.

pH Inhibition

Menu Location: **Project|Parameters|Kinetic|pH**

Name	Default Value	Unit	Explanation
Heterotrophs low pH limit (anaerobic)	5.5	pH units	At a pH equal to this value the growth rate of ordinary heterotrophic biomass in an anaerobic will be reduced by 50%.
Heterotrophs high pH limit (anaerobic)	8.5	pH units	At a pH equal to this value the growth rate of ordinary heterotrophic biomass in an anaerobic will be reduced by 50%.

Switching Functions

Menu Location: **Project|Parameters|Kinetic|Switches**

Name	Default Value	Unit	Explanation
Heterotrophic DO half sat.	0.05	mgO_2/L	This constant is used to switch off aerobic OHO activity under low DO conditions (that is in anaerobic and anoxic reactors).
Anoxic NO3 half sat.	0.1	mgN/L	This constant is used to switch off anoxic growth processes using nitrate under low nitrate conditions.
Anoxic NO2 half sat.	0.1	mgN/L	This constant is used to switch off anoxic growth processes using nitrite under low nitrite conditions.
NH3 nutrient half sat.	0.001	mgN/L	This constant is used to slow all biomass growth processes at low ammonia-N concentrations (N nutrient limiting.
P nutrient half sat.	0.001	mgP/L	This constant is used to slow the growth of biomass when there is no phosphorus available as nutrient.
Heterotrophic Hydrogen half sat.	1.0	mgCOD/L	This constant switches between two fermentation pathways, generating acetate and propionate in various ratios, depending on available H_2 concentration.
Synthesis anion/cation half sat.	0.01	meq/L	Half saturation concentration for anions and cations.

Growth and Decay of Propionic Acetogens

Number of Processes: 2

Engineering Objective: anaerobic digestion

Implementation: permanent, always active in the BioWin model

Module Description:

These 2 processes describe the growth and decay of propionic acetogens, converting propionate to acetate, CO_2 and hydrogen. Nitrogen source for cell synthesis is ammonia. The base rate expression the growth process is the product of the maximum specific growth rate, the propionic acetogen biomass concentration and a Monod expression for propionate. This base rate is modified to account for environmental conditions (off unless anaerobic, inhibited by hydrogen and acetate), nutrient limitations (nitrogen, phosphate, other cations and anions) and pH inhibition. BioWin uses ammonia as a nitrogen source for cell synthesis.

The decay process has a rate that varies according to the electron acceptor environment

Model parameters are listed in:

Kinetic Parameters

Menu Location: **Project|Parameters|Kinetic|Acetogens**

Name	Default Value	Unit	Explanation
Max. spec. growth rate	0.25	d^{-1}	Maximum specific growth rate in the absence of substrate limitations.
Substrate half sat.	10.0	mgCOD/L	Half saturation for regulation of growth rate, based on availability of propionate as substrate
Acetate inhibition	10000	mgCOD/L	Acetate inhibition constant: high acetate concentrations inhibit propionic acetogen growth.
Decay rate	0.05	d^{-1}	Decay rate constant when there is no oxygen available.
Aerobic decay rate	0.52	d^{-1}	Decay rate constant in the presence of oxygen.

Stoichiometric

Menu Location: **Project|Parameters|Stoichiometric|Acetogens**

Name	Default Value	Unit	Explanation
Yield	0.1	mgCOD/mgCOD	Amount of biomass produced on one unit of propionate converted.
H2 yield	0.4	mgCOD/mgCOD	Amount of H_2 produced on one unit of propionate converted.
CO2 yield	1.0	mmolCO_2/mmol	Moles of CO_2 produced per mole
N in biomass	0.07	mgN/ mgCOD	N content of propionic acetogens.
N in endogenous	0.07	mgN/ mgCOD	N content of endogenous residue originating from

residue			propionic acetogen decay.
P in biomass	0.022	mgP/ mgCOD	P content of propionic acetogens.
P in endogenous residue	0.022	mgP/ mgCOD	P content of endogenous residue originating from propionic acetogen decay.
Fraction to endogenous residue	0.08	-	Fraction of biomass that becomes inert upon decay.
COD:VSS ratio	1.42	mgVSS/mgCOD	Conversion factor between biomass as measured in COD and its VSS content. This value is relatively stable for biomass.

pH Inhibition

Menu Location: **Project|Parameters|Kinetic|pH**

Name	Default Value	Unit	Explanation
Propionic acetogens low pH limit	4.0	pH units	At a pH equal to this value the growth rate of propionic acetogens will be reduced by 50%.
Propionic acetogens high pH limit	10.0	pH units	At a pH equal to this value the growth rate of propionic acetogens will be reduced by 50%.

Switching Functions

Menu Location: **Project|Parameters|Kinetic|Switches**

Name	Default Value	Unit	Explanation
Heterotrophic DO half sat.	0.05	mgO_2/L	This constant is used to switch off aerobic OHO activity under low DO conditions (that is in anaerobic and anoxic reactors).
Anoxic NO3 half sat.	0.1	mgN/L	This constant is used to switch off anoxic growth processes using nitrate under low nitrate conditions.
Anoxic NO2 half sat.	0.1	mgN/L	This constant is used to switch off anoxic growth processes using nitrite under low nitrite conditions.
NH3 nutrient half sat.	0.001	mgN/L	This constant is used to slow all biomass growth processes at low ammonia-N concentrations (N nutrient limiting.
P nutrient half sat.	0.001	mgP/L	This constant is used to slow the growth of biomass when there is no phosphorus

			available as nutrient.
Propionic acetogens Hydrogen limit	5.0	mgCOD/L	This constant is used to inhibit the growth of biomass when high levels of H_2 are present.
Synthesis anion/cation half sat.	0.01	meq/L	Half saturation concentration for anions and cations.

Growth and Decay of Methanogens

Number of Processes: 6

Engineering Objective: anaerobic digestion

Implementation: permanent, always active in the BioWin model

Module Description:

These 6 processes describe the growth and decay of two of the principal groups of obligate anaerobic microorganisms (acetoclastic methanogens converting acetate (or methanol) to methane and CO_2; and hydrogenotrophic methanogens, converting CO_2 (or methanol) and hydrogen to methane and water).

The base rate expression for each of the 4 growth processes is the product of the maximum specific growth rate constant, the appropriate biomass concentration and a Monod expression for each of the substrates. This base rate is modified to account for nutrient limitations (ammonia, phosphate, other cations and anions) and pH inhibition. BioWin uses ammonia as a nitrogen source for cell synthesis

For both populations, the decay rate that varies according to the electron acceptor environment.

Model parameters are listed in:

Kinetic Parameters

Menu Location: **Project|Parameters|Kinetic|Methanogens**

Name	Default Value	Unit	Explanation
Acetoclastic Mu Max	0.3	d^{-1}	Maximum specific growth rate for the acetoclastic biomass if no substrate limitation or inhibition occurs.
H2-utilizing Mu Max	1.4	d^{-1}	Maximum specific growth rate for the H_2-utilizing biomass if no substrate limitation or inhibition occurs.
Acetoclastic Ks	100	mgCOD/L	Half saturation for regulation of acetoclastic biomass growth rate, based on availability of acetate as substrate.
Acetoclastic	0.5	mgCOD/L	Half saturation concentration of

methanol Ks			methanol for acetoclastic biomass.
Hydrogenotrophic CO2 half sat.	0.1	mmolCO_2/L	Half saturation for regulation of H_2-utilizing biomass growth rate, based on availability of CO_2 as substrate.
H2-utilizing Ks	0.1	mgCOD/L	Half saturation for regulation of H_2-utilizing biomass growth rate, based on availability of hydrogen as substrate.
H2-utilizing methanol Ks	0.5	mgCOD/L	Half saturation concentration of methanol for H_2-utilizing biomass.
Acetoclastic propionic inhibition	10000	mgCOD/L	Propionate inhibition constant: high levels of propionate inhibit acetoclastic biomass growth.
Acetoclastic decay rate	0.13	d^{-1}	Decay rate constant when there is no oxygen available.
Acetoclastic aerobic decay rate	0.6	d^{-1}	Decay rate constant in the presence of oxygen.
H2-utilizing decay rate	0.13	d^{-1}	Decay rate constant when there is no oxygen available.
H2-utilizing aerobic decay rate	0.6	d^{-1}	Decay rate constant in the presence of oxygen.

Stoichiometric

Menu Location: **Project|Parameters|Stoichiometric|Methanogens**

Name	Default Value	Unit	Explanation
Acetoclastic yield	0.1	mgCOD/mgCOD	Amount of acetoclastic biomass produced using one unit of substrate (acetate). The rest of the substrate will be converted to CO_2.
Methanol acetoclastic yield	0.1	mgCOD/mgCOD	Acetoclastic biomass yield on one unit of methanol COD.
H2-utilizing yield	0.1	mgCOD/mgCOD	Amount of H_2-utilizing biomass produced using one unit of substrate (hydrogen). The rest of the substrate will be converted to methane and water.
Methanol H2-utilizing yield	0.1	mgCOD/mgCOD	H_2-utilizing biomass yield on one unit of methanol COD
N in acetoclastic biomass	0.07	mgN/mgCOD	N content of acetoclastic biomass.
N in H2-utilizing biomass	0.07	mgN/mgCOD	N content of H_2-utilizing biomass.
N in acetoclastic endog. residue	0.07	mgN/mgCOD	N content of endogenous residue originating from acetoclastic decay.

H2-utilizing N in endog. Residue	0.07	mgN/mgCOD	N content of endogenous residue originating from H_2-utilizing biomass decay.
P in acetoclastic biomass	0.022	mgP/mgCOD	P content of acetoclastic biomass.
P in H2-utilizing biomass	0.022	mgP/mgCOD	P content of H_2-utilizing biomass.
P in acetoclastic endog. residue	0.022	mgP/mgCOD	P content of endogenous residue originating from acetoclastic decay.
P in H2-utilizing endog. Residue	0.022	mgP/mgCOD	P content of endogenous residue originating H_2-utilizing biomass.
Acetoclastic fraction to endog. residue	0.08	-	Fraction of acetoclastic biomass that becomes inert upon decay.
H2-utilizing fraction to endog. residue	0.08	-	Fraction of H_2-utilizing biomass that becomes inert upon decay.
Acetoclastic COD:VSS ratio	1.42	mgCOD/mgVSS	Conversion factor between biomass as measured in COD and its VSS content. This value is relatively stable for biomass.
H2-utilizing COD:VSS ratio	1.42	mgCOD/mgVSS	Conversion factor between biomass as measured in COD and its VSS content. This value is relatively stable for biomass.

pH Inhibition

Menu Location: **Project|Parameters|Kinetic|pH**

Name	Default Value	Unit	Explanation
Acetoclastic methanogens low pH limit	5	pH units	At a pH equal to this value the growth rate of the methanogens will be reduced by 50%. Methanogens are sensitive to low pH, the digester can easily turn acid.
Acetoclastic methanogens high pH limit	9	pH units	At a pH equal to this value the growth rate of the methanogens will be reduced by 50%
H2-utilizing methanogens low pH limit	5	pH units	At a pH equal to this value the growth rate of the methanogens will be reduced by 50%. Methanogens are sensitive to low pH, the digester can easily turn acid.
H2-utilizing methanogens high pH limit	9	pH units	At a pH equal to this value the growth rate of the methanogens will be reduced by 50%

Switching Functions

Menu Location: **Project|Parameters|Kinetic|Switches**

Name	Default Value	Unit	Explanation

Name	Default Value	Unit	Explanation
Heterotrophic DO half sat.	0.05	mgO_2/L	This constant is used to switch off aerobic OHO activity under low DO conditions (that is in anaerobic and anoxic reactors).
Anoxic NO3 half sat.	0.1	mgN/L	This constant is used to switch off anoxic growth processes using nitrate under low nitrate conditions.
Anoxic NO2 half sat.	0.1	mgN/L	This constant is used to switch off anoxic growth processes using nitrite under low nitrite conditions.
NH3 nutrient half sat.	0.001	mgN/L	This constant is used to slow all biomass growth processes at low ammonia-N concentrations (N nutrient limiting.
P nutrient half sat.	0.001	mgP/L	This constant is used to slow the growth of biomass when there is no phosphorus available as nutrient.
Synthesis anion/cation half sat.	0.01	meq/L	Half saturation concentration for anions and cations.

Chemical Precipitation Model

The Chemical Precipitation Model in BioWin describes insoluble metal salt formation that occurs in wastewater treatment plants under various environmental conditions. The objective of this model is to improve the prediction of soluble phosphate residuals and provide accurate chemical sludge production estimates. Phosphorus plays an active part both in biological and chemical processes in wastewater. It is an important part of the weak acid-base system, it is a nutrient and an energy-storage compound for various microorganisms and it readily forms insoluble precipitates with magnesium, calcium, as well as iron and aluminum ions if those are added to the wastewater. All these processes need to be taken into account for accurate effluent phosphorus predictions.

The Chemical Precipitation Model in BioWin contains the following functional categories, modules

Chemical Phosphorus Precipitation by Alum or Ferric

Struvite and Calcium Phosphates Precipitation

These modules are described in detail below.

Ferric or Alum Precipitation

Number of Reactions: 6

Engineering Objective: Chemical phosphorus removal

Implementation: Optional, has to be activated in the BioWin model options

Module Description:

In the current implementation either Ferric or Aluminum phosphate precipitation can be selected as an option, but not both. Since precipitation is orders of magnitude faster than biological reactions, the model equations are expressed and solved using an equilibrium approach. The added metal will form an insoluble phosphate/hydroxo complex ($Fe_{1.6}H_2PO_4OH_{3.8}$ or $Al_{0.8}H_2PO_4OH_{1.4}$), a soluble metal-phosphate complex ($FeH_2PO_4^{2+}$ or $AlHPO_4^{+}$), and any residual metal will be mostly bound in metal hydroxide precipitate ($Fe(OH)_3$ or $Al(OH)_3$). In equilibrium there is always a low concentration of free metal ions and soluble phosphate species in various dissociated or undissociated forms (PO_4^{3-}, HPO_4^{2-}, $H_2PO_4^{-}$, H_3PO_4). The phosphate species together with the soluble metal-phosphate complex cause the residual, soluble effluent phosphorus concentration. The reactions are handled using the proper solubility and dissociation equations. The distribution and residual concentration of these components is pH and dose dependent. For soluble phosphorus species in case of a large metal overdose the following graphs show the pH sensitivity:

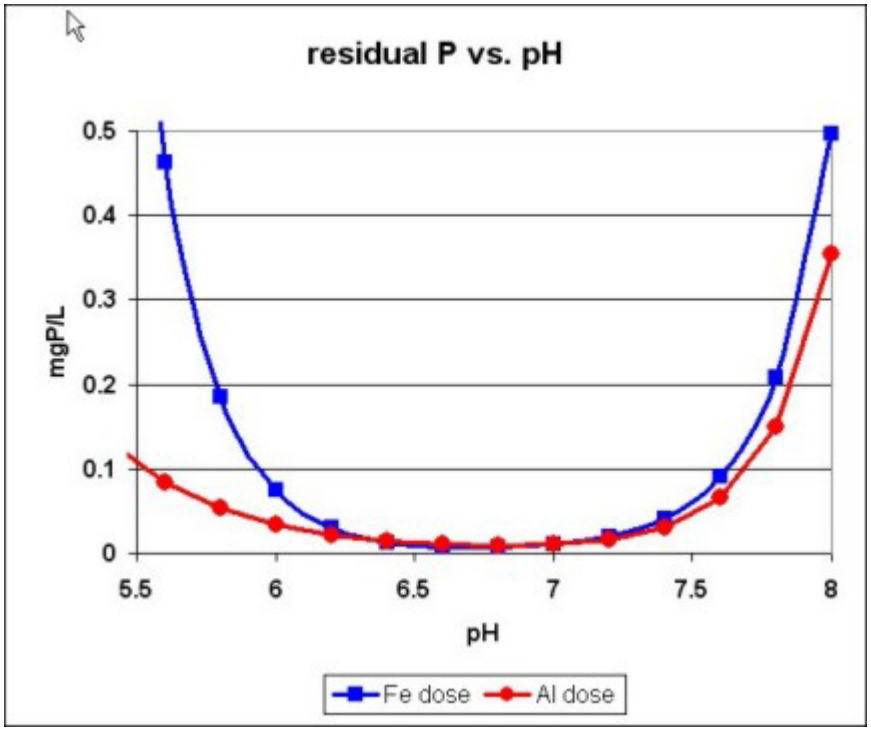


Figure 1 : pH sensitivity for soluble phosphorus species in case of a large metal overdose

Model parameters are listed in:

Menu Location: **Project|Parameters|Other|Physico-chemical constants**

Name	Default Value	Unit	Explanation

Equilibrium soluble PO4 with Al dosing at pH 7	0.01	mgP/L	This is the best achievable soluble PO4 concentration when the solution is in equilibrium with the composite product of $Al_rH_2PO_4OH_{(3r-1)}$ based on the following reaction equation: $\left(Al^{3+}\right)^{0.8}\left(H_2PO_4^-\right)\left(OH^-\right)^{1.4} = K_{sp,AlP}$ $\left(Al^{3+}\right)^{0.8}\left(H_2PO_4^-\right)\left(OH^-\right)^{1.4} = K_{sp,AlP}$
Al to P mole ratio (r) in $Alr(H_2PO_4)(OH)_3r\text{-}1$ precipitation	0.8	mmolAl/mmolP	Under low doses and optimal pH this is the default molar ratio between the precipitated aluminum and phosphate ions. However the actual (observed) ratio will depend on pH, as well as the formation of other aluminum phosphate and hydroxide components.
$Al(OH)_3$ solubility product	1.2590E+9	mol/L	This solubility constant is based on the reaction $Al^{3+} + 3H_2O \leftrightarrow Al(OH)_3 + 3H^+$, as expressed by: $\frac{\left(Al^{3+}\right)}{\left(H^+\right)^3} = K_{sp,AlOH3}$ The value of this constant should never be changed. The hydroxide precipitate formed in this reaction uses up aluminum ions and contributes to the non-stoichiometric Al/P ratios observed as well as the required overdose.
$AlHPO_4$+ dissociation constant	7.9430E-13	mol/L	This dissociation constant is based on the following reaction equation: $\frac{\left(Al^{3+}\right)\left(HPO_4^{2-}\right)}{\left(AlHPO_4^+\right)} = K_{iAlHPO4}$ This soluble aluminum phosphate complex will contribute to the residual soluble phosphate concentration measured.
Equilibrium soluble PO4 with Fe dosing at pH 7	0.01	mgP/L	This is the best achievable soluble PO4 concentration when the solution is in equilibrium with the composite product of $Fe_{1.6}H_2PO_4OH_{3.8}$ based on the following reaction equation: $\left(Fe^{3+}\right)^r\left(H_2PO_4^-\right)\left(OH^-\right)^{3r-1} = K_{sp,FeP}$
Fe to P mole ratio (r) in $Fer(H_2PO_4)(OH)_3r\text{-}1$ precipitation	1.6	mmolFe/mmolP	Under low doses and optimal pH this is the default molar ratio between the precipitated ferric and phosphate ions. However the actual (observed) ratio will depend on pH, as well as the formation of other ferric phosphate and hydroxide components.
$Fe(OH)_3$ solubility product	0.05	mol/L	This solubility constant is based on the reaction: $Fe^{3+} + 3H_2O \leftrightarrow Fe(OH)_3 + 3H^+$, as expressed by:

			$\frac{(Fe^{3+})}{(H^+)^3} = K_{sp,FeOH3}$ The value of this constant should never be changed. The hydroxide precipitate formed in this reaction uses up ferric ions and contributes to the non-stoichiometric Fe/P ratios observed as well as the required overdose.
$FeH2PO_4$++ dissociation constant	5.0120E-22	mol/L	This dissociation constant is based on the following reaction equation: $\frac{(Fe^{3+})(H_2PO_4^-)}{(FeH_2PO_4^{2+})} = K_{iFeH_2PO_4}$ This soluble ferric phosphate complex will contribute to the residual soluble phosphate concentration measured.

Spontaneous Chemical Precipitation

Module Description:

Magnesium and Calcium is present in most wastewaters and can spontaneously form precipitates. From the large number of phosphate and carbonate precipitates that can be formed under typical conditions, particularly under higher than neutral pH, the most important ones affecting soluble phosphorus levels are struvite (magnesium-ammonium-phosphate, $MgNH_4PO_4 \cdot 6H_2O$), and hydroxy-dicalcium-phosphate (HDP, $Ca_2HPO_4(OH)_2$).

Struvite typically forms in digesters, due to the high concentration of magnesium, ammonium and phosphate ions present. These ions are transported to the digester in the biomass and are released. BioWin (in the anaerobic module) describes Mg, N, and P release. The resulting struvite precipitation can occur particularly in pipes and on overflow weirs, where degassing of CO_2 raises the pH. The model is described in detail in Musvoto et al. (2000).

HDP (and many other types of calcium phosphates) can precipitate in the bioreactor due to pH changes occurring in the anaerobic/aerobic zones. Accurate prediction of phosphate levels in bio-P processes may require simulating the HDP precipitation – redissolution phenomena as well. BioWin contains one more calcium phosphate precipitate, hydroxy-apatite (HAP), that is considered a sink for calcium that does not redissolve. The model is described in Maurer et al. (1999).

These precipitation processes are formulated with kinetic equations, according to the referenced papers.

Number of Processes: 3

Engineering Objective: Formation of Struvite and Calcium Phosphates

Implementation: Optional, has to be activated in the BioWin model options

Model parameters are listed in:

Menu Location: **Project|Parameters|Other|Physico-chemical rates**

Name	Default Value	Unit	Explanation

Struvite precipitation rate	3.0000E+10	d^{-1}	The published precipitation rate (Musvoto et al. expressed in molar terms) is $3*10^{15}$. Simulation of other experiments suggests a much lower rate (Suzuki 1998). Mixing effects likely have an influence on the rate constant. The default selected is a compromise and can be changed in the range of 10^{9} to $3*10^{15}$.
Struvite redissolution rate	3.0000E+11	d^{-1}	This constant has a lower importance as struvite redissolution is not typically encountered in wastewater processes.
Struvite half sat.	1.0	mgTSS/L	This is an empirical constant designed to provide continuity to the mathematical solution by maintaining low levels of struvite even under lower pH conditions. Its value should not be changed.
HDP precipitation rate	1.0000E+8	L/(molP.d)	The value of this rate constant will affect how close to equilibrium the HDP precipitation reaction will be under the given hydraulic resident times. If it is necessary to change it, it should be changed together with the redissolution rate constant.
HDP redissolution rate	1.0000E+8	L/(molP.d)	The value of this rate constant will affect how close to equilibrium the HDP redissolution reaction will be under the given hydraulic residence times. If it is necessary to change it, it should be changed together with the precipitation rate constant.
HAP precipitation rate	5.0000E-4	molHDP/(L.d)	This constant rate will generate insoluble HAP that cannot be redissolved.

Menu Location: **Project|Parameters|Other|Physico-chemical constants**

Name	Default Value	Unit	Explanation
Struvite solubility product	6.9180E-14	mol/L	This solubility constant should not be changed
HDP solubility product	2.7500E-22	mol/L	This solubility constant should not be changed
HDP half sat.	1.0	mgTSS/L	This is an empirical constant designed to provide continuity to the mathematical solution by maintaining low levels of

			HDP even under lower pH conditions. Its value should not be changed.

Settling Models

Engineering Objective: Gravitational solid / liquid separation

Implementation: Flux-based solids / liquid separation models (used in one-dimensional model clarifier elements)

Module Description:

The general flux theory approach models solids settling in one dimension. In the one-dimensional approach solids and liquid movement in the vertical direction are assumed to be dominant and horizontal movement is ignored. The settling tank is divided into a number of layers in the vertical direction and a numerical technique is used to solve the mass balance equations in the vertical direction. The solution to the mass balance equations provides the solids concentration profile in the settling tank, and the solids concentration in the effluent and underflow. For more details, please see the **Flux Based Models** section of the "*Settling and Solid / Liquid Separation Models*" chapter.

BioWin offers two types of flux theory models:

1 – Modified Vesilind

2 – Double Exponential

Each model has its own set of parameters; these are described below.

Modified Vesilind Parameters

Menu Location: **Project|Parameters|Settling|Modified Vesilind**

Name	Default Value	Unit	Explanation
Maximum Vesilind settling velocity (V_0)	170	m/d	The maximum attainable settling velocity at theoretically infinite dilution in the unmodified Vesilind relationship. Primarily affects clarification function. Higher values are associated with well-settling sludge.
Vesilind hindered zone settling parameter (K)	0.37	L/g	The term which describes the exponential decrease in settling velocity with increasing concentration in the Vesilind relationship. Primarily affects the thickening function. Higher values associated with poor settling sludge.
Clarification switching function	100	mg/L	Switch applied in a Monod-type switching function to the Vesilind relationship. Addresses unrealistically high settling velocities predicted at low solids concentrations by strict application of the Vesilind model as originally published. Increasing the value for this switch will result in higher

			effluent suspended solids predictions.
Specified TSS conc.for height calc.	2500	mg/L	BioWin will locate and plot the height of the specified suspended solids concentration within the settler profile.
Maximum compactability constant	15,000	mg/L	The specified value is used to limit the maximum suspended solids concentration that can be achieved in a model settler layer. Also, as the solids concentration in a layer approaches this concentration, resuspension of solid particles occurs.

Double Exponential Parameters

Menu Location: **Project|Parameters|Settling|Double Exponential**

Name	Default Value	Unit	Explanation
Maximum Vesilind settling velocity (V_0)	410	m/d	The maximum attainable settling velocity at theoretically infinite dilution in the unmodified Vesilind relationship.
Maximum (practical) settling velocity (V_0')	270	m/d	The maximum settling velocity attainable in the Double Exponential model. This constraint addresses the unrealistically high settling velocities predicted at low solids concentrations by strict application of the Vesilind model. Higher values are associated with well-settling sludge.
Hindered zone settling parameter (Kh)	0.40	L/g	The term which describes the exponential decrease in settling velocity with increasing concentration in Zone 4 of the Double Exponential model where hindered settling dominates. Higher values associated with poor settling sludge.
Flocculent zone settling parameter (Kf)	2.5	L/g	The exponential term which describes settling behavior in Zone 2 of the Double Exponential model where flocculant settling is dominant. Higher values associated with poor settling sludge.
Maximum non-settleable TSS	20	mg/L	The minimum attainable suspended solids concentration in a layer is calculated as a fraction of the settling tank influent solids concentration, but it may not be less than this user-specified value.
Non-settleable fraction	0.001	-	The minimum attainable suspended solids concentration in a layer is calculated as a fraction of the settling tank influent solids concentration.
Specified TSS conc. for height calc.	2500	mg/L	BioWin will locate and plot the height of the specified suspended solids concentration within the settler profile.

pH Model

The **pH model** is described in detail in the *"Modeling of pH in BioWin"* chapter.

Aeration and Gas Transfer Model

Parameters used in BioWin's aeration and gas transfer model are accessed via a number of different tabs. These are listed in the following sections.

Number of Processes: 6

Engineering Objective: Gas-liquid mass transfer

Implementation: permanent, always active in the BioWin model

Module Description:

There are six gas-liquid mass transfer processes to allow interphase transfer of oxygen, carbon-dioxide, methane, nitrogen, ammonia and hydrogen. Details about the mass transfer model may be found in the *"Gas-Liquid Mass Transfer"* chapter.

The Mass transfer model is impacted by the values of the following model parameters.

Mass transfer Parameters

Menu Location: **Project|Parameters|Other|Mass transfer**

Name	Default Value	Unit	Explanation
K_L for H_2	17.0	m/d	Liquid phase mass transfer coefficient for H_2
K_L for CO_2	10.0	m/d	Liquid phase mass transfer coefficient for CO_2
K_L for NH_3	1.0	m/d	Liquid phase mass transfer coefficient for NH_3
K_L for CH_4	8.0	m/d	Liquid phase mass transfer coefficient for CH_4
K_L for N_2	15.0	m/d	Liquid phase mass transfer coefficient for N_2
K_L for O_2	13.0	m/d	Liquid phase mass transfer coefficient for O_2

Aeration Parameters

Menu Location: **Project|Parameters|Other|Aeration**

Name	Default Value	Unit	Explanation
Alpha (surf) OR Alpha F (diff)	0.50	Unitless	α is the ratio of the overall mass transfer coefficient in process water to the overall mass transfer coefficient in clean water. F (diffuser fouling factor) is the ratio of the overall mass transfer coefficient for a particular diffuser after a given time in service to that of a new diffuser in the same process water.

Beta	0.95	Unitless	β is the ratio of the dissolved oxygen saturation concentration in process water to the saturation concentration in clean water
Surface pressure	101.325	kPa	Atmospheric pressure at field conditions.
Fractional effective saturation depth (Fed)	0.325	Unitless	The effective saturation depth is the depth at which the total pressure (hydrostatic and atmospheric) would produce a saturation concentration equal to the steady state saturation concentration for the system.
Supply gas CO2	0.035	vol. %	The volume percentage of carbon dioxide in the supply gas to a diffused air system. This parameter is also used to determine the dissolved carbon dioxide saturation concentration for surface aerator systems.
Supply gas O2	20.95	vol. %	The volume percentage of oxygen in the supply gas to a diffused air system. This parameter is also used to determine the dissolved oxygen saturation concentration for surface aerator systems.
Off-gas CO2	2.0	vol. %	The volume percentage of carbon dioxide in the gas leaving a diffused air system. [This parameter is not used if the gas phase is modeled.]
Off-gas O2	18.8	vol. %	The volume percentage of oxygen in the gas leaving a diffused air system. [This parameter is not used if the gas phase is modeled.]
Off-gas H2	0.00	vol. %	The volume percentage of hydrogen in the gas leaving a diffused air system. [This parameter is not used if the gas phase is modeled.]
Off-gas NH3	0.00	vol. %	The volume percentage of ammonia in the gas leaving a diffused air system. [This parameter is not used if the gas phase is modeled.]
Surface turbulence factor	2	Unitless	This parameter indicated the intensity of mixing on the surface conditions (it has little impact in aerated systems).
Set point controller gain	1.0	Unitless	This parameter may be used to increase the gain of the dissolved oxygen controller. Typically the user would increase this value if the controller was slow in achieving the dissolved oxygen set point.

Diffuser Parameters

Location: '**Model Parameters' on Element "Operation" tab**

Name	Default Value	Unit	Explanation
k_1 in $C = k_1(PC)^{0.25}$	2.5656		Correlation parameter

$+ k_2$			
k_2 in $C = k_1(PC)^{0.25} + k_2$	0.04320		Correlation parameter
Y in $K_La = C\ Usg^Y$	0.82000		Correlation parameter
Area of one diffuser	0.04100	m^2	Area of a single diffuser is required to determine the number of diffusers.
% of tank area covered by diffusers (PC)	10.00000	%	Ratio of the total active diffuser area to the tank area as a percentage.
Diffuser mounting height	0.25	m	Height of diffuser discharge above tank floor
Min. air flow rate per diffuser	0.50000	$m^3\ hr^{-1}$	Minimum of X axis in SOTE plot.
Max. air flow rate per diffuser	10.00	$m^3\ hr^{-1}$	Maximum of X axis in SOTE plot.

Surface aerator Parameters

Location: '**Model Parameters' on Element "Operation" tab**

Name	Default Value	Unit	Explanation
Surface aerator Std. oxygen transfer rate	1.500000	kg O $kW^{-1}\ hr^{-1}$	Standard oxygen transfer rate for rotary surface aerators
Maximum power per rotor	20.00000	kW	The maximum power input per surface aerator is used to determine the number of aerators required.

Model Library for the Model Builder

The following builder models have been included with BioWin:

- ASM1
- ASM2d
- ASM3
- Aerobic growth of methylotrophs on methanol
- Inert conversion
- $CaCO_3$ precipitation

For more information on loading these models, and using the Model Builder interface in general, please see the **Model Builder** section of the *"Useful BioWin Interface Tools and Techniques"*chapter.

The following points should be noted when using these Builder models:

- **OURc** – the displayed value for this parameter is the carbonaceous OUR for the BioWin model only. Therefore when a Model Builder model is active, oxygen demand from any user-defined processes are not included (OURn is similar but only applies to the oxygen utilized for AOB and NOB growth). In summary, OURc and OURn only reflect the BioWin model terms; if there is no BioWin model used in the Model Builder then they will be zero.
- **OURt** is calculated from the sum of all process rates that use or produce oxygen. Consequently if the BioWin reactions are ON in the model builder then:
- **OURt - OURn - OURc** = the **OUR** from the Model Builder reactions
- and if BioWin reactions are **Off** in the **Model Builder** then:
- **OURt** = the **OUR** from the Model Builder reactions.

Definition of Combined Variables

To model the range of system types it is necessary to track the concentrations of a large number of components (state variables) in each stream of a configuration. In certain cases tracking a number of the variables may be superfluous. For example, if a system incorporates only an aerobic activated sludge unit then all variables relating to biological phosphorus removal will have zero concentration values (polyP heterotrophic organism mass, stored polyphosphate, etc.).

Each element in the configuration is modeled to provide a mass balance on the following state variables.

Short name	Full name	Units
Zbh	Non-polyP heterotrophs	[mgCOD/L]
Zbmeth	Anoxic methanol utilizers	[mgCOD/L]
Zaob	Ammonia oxidizing biomass	[mgCOD/L]
Znob	Nitrite oxidizing biomass	[mgCOD/L]
Zamox	Anaerobic ammonia oxidizers	[mgCOD/L]
Zbp	PolyP heterotrophs	[mgCOD/L]
Zbpa	Propionic acetogens	[mgCOD/L]
Zbam	Acetoclastic methanogens	[mgCOD/L]
Zbhm	Hydrogenotrophic methanogens	[mgCOD/L]
Ze	Endogenous products	[mgCOD/L]
Xsp	Slowly bio. COD (part.)	[mgCOD/L]
Xsc	Slowly bio. COD (colloid.)	[mgCOD/L]
Xi	Part. inert. COD	[mgCOD/L]
Xon	Part. bio. org. N	[mgN/L]
Xop	Part. bio. org. P	[mgP/L]
Xin	Part. inert N	[mgN/L]

Xip	Part. inert P	[mgP/L]
Spha	Stored PHA	[mgCOD/L]
PP-lo	Releasable stored polyP	[mgP/L]
PP-hi	Fixed stored polyP	[mgP/L]
XPPCat	PolyP bound cations	[mg/L]
Sbsc	Readily bio. COD (complex)	[mgCOD/L]
Sbsa	Acetate	[mgCOD/L]
Sbsp	Propionate	[mgCOD/L]
Sbmeth	Methanol	[mgCOD/L]
SbH2	Dissolved H2	[mgCOD/L]
CH4	Dissolved methane	[mg/L]
NH3-N	Ammonia N	[mgN/L]
Nos	Sol. bio. org. N	[mgN/L]
NO2-N	Nitrite N	[mgN/L]
NO3-N	Nitrate N	[mgN/L]
N2	Dissolved nitrogen gas	[mgN/L]
PO4-P (incl. MeP)	PO4-P (Sol. & Me Complexed)	[mgP/L]
Sus	Sol. inert COD	[mgCOD/L]
Nus	Sol. inert TKN	[mgN/L]
ISS	Inorganic S.S.	[mgTSS/L]
XStru	Struvite	[mgTSS/L]
XHDP	Hydroxy-dicalcium-phosphate	[mgTSS/L]
XHAP	Hydroxy-apatite	[mgTSS/L]
SMg	Magnesium	[mg/L]
SCa	Calcium	[mg/L]
Me	Metal	[mg/L]
SCat	Other Cations (strong bases)	[meq/L]
SAn	Other Anions (strong acids)	[meq/L]
SCO2	Total CO2 [mmol/L]	[mmol/L]
UD1	User defined 1	[mg/L]
UD2	User defined 2	[mg/L]
UD3	User defined 3 [mgVSS/L]	[mgVSS/L]
UD4	User defined 4	[mgTSS/L]
DO	Dissolved oxygen	[mg/L]
Flow	Flow	Unit system dependant
Liq.Vol.	Liquid volume	Unit system dependant
Temp.	Temperature	Degrees celcius

BioWin calculates a number of combined variables that may be displayed for certain elements. The following table outlines these combined variables.

Short name	Full name	Description

VSS	Volatile suspended solids	VSS is calculated converting all particulate COD state variables (biomasses, substrates and inert organics) with their proper COD/VSS ratios to VSS, adding User Defined Variable #3 and summing them. Biomasses have their own Fcv (COD/VSS) conversion factors, while for other components the ratios indicated in the **Project|Parameters|Other|General** dialog are used. The VSS is calculated as follows: VASS + Ze/$Fcv_{,ZBH}$ + Xsp/$Fcv_{,XSP}$ + Xi/$Fcv_{,XI}$ + Sphb/$Fcv_{,XSP}$ +UD3
VASS	Volatile active suspended solids	VASS is calculated converting all biomass state variables with their proper COD/VSS ratios to VSS, and summing them. Biomasses have their own Fcv (COD/VSS) conversion factor in the proper stoichiometric parameter forms. The VASS is calculated as follows: Zbh/$Fcv_{,ZBH}$+ Zbmeth/$Fcv_{,ZBMETH}$ + Zbaob/$Fcv_{,ZBAOB}$ + Zbnob/$Fcv_{,ZBNOB}$ + Zbanammox/$Fcv_{,ZBANAMMOX}$ + Zbp/$Fcv_{,ZBP}$ + Zbpa/$Fcv_{,ZBPA}$ + Zbam/$Fcv_{,ZBAM}$ + Zbhm/$Fcv_{,ZBHM}$
TISS	Total inorganic suspended solids	TISS is the sum of inorganic solids from the influent (ISS), total precipitated solids, the poly-P content, UD4, and the organism ash content; that is: XPrec + ISS + PP-lo + PP-hi +XPPCat + UD4 + (VASS + Ze/$Fcv_{,ZBH}$)*(1/(1-AshContent/100.0) - 1.0)
TSS	Total suspended solids	TSS is calculated as a sum of VSS and TISS.
XCOD	Particulate COD	Particulate COD is calculated as the sum of all particulate state variables that are expressed in COD, as follows: Zbh + Zbmeth + Zba + Zbp + Zbpa + Zbam + Zbhm + Ze + Xsp + Xi +Sphb
SCOD	Filtered COD	Filtered COD is calculated as the sum of all soluble state variables that are expressed in COD, including the colloids and hydrogen as follows: Sbsc + Sbsa + Sbsp + SbMeth + Sh2 + Sus + Xsc
COD	Total Chemical Oxygen Demand	Total COD is the sum of filtered and particulate COD.
SPO_4	Soluble PO_4-P	Soluble PO_4-P is the sum of all inorganic phosphate species, including soluble metal phosphate complexes in case of metal dosage. Note that biodegradable and unbiodegradable soluble and colloidal organic P is considered zero. That is: (SolubleP & Metal-complexed P) - ($Me_rH_2PO_4OH_{3r-1}$)
TP	Total P	Total P is calculated as the sum of Soluble PO_4-P, P content of biomasses and endogenous residue as a fraction, particulate inert and biodegradable organic P, P in releasable and fixed polyphosphate, and P bound in Struvite, HDP, HAP and metal phosphate precipitate if applicable. That is: Zbh*$Fzbp_{,ZBH}$ + Zbmeth*$Fzbp_{,ZBMETH}$ + Zbaob*$Fzbp_{,ZBAOB}$+ Zbnob*$Fzbp_{,ZBNOB}$+ Zbanammox*$Fzbp_{,ZBANAMMOX}$ + Zbp*$Fzbp_{,ZBP}$ + Zbpa*$Fzbp_{,ZBPA}$ + Zbam*$Fzbp_{,ZBAM}$ + Zbhm* $Fzbp_{,ZBHM}$ + PP-lo + PP-hi + Xip + (SolubleP &

		Metal-complexed P) + Xop + Xstru *(MWPhosphorus/MWStru) + XHDP*(MWPhosphorus/MWXHDP) + XHAP*(3*MWPhosphorus)/MWXHAP
XPrec	Total precipitated solids	Precipitated solids are calculated as the sum of HDP, HAP, Struvite, Metal phosphate and metal hydroxide solids, that is: XHDP + XHAP + Xstru + $MeOH_3$+ $Me_rH_2PO_4OH_{3r-1}$
STKN	Filtered TKN	Filtered TKN is ammonia-N, and soluble biodegradable and unbiodegradable organic nitrogen, that is: NH3 + Nos + Nus
XTKN	Particulate TKN	Particulate TKN is calculated as the sum of the N content of biomasses, particulate inert and biodegradable organic N, and the N content of struvite; that is: Zbh*Fzbn,$_{ZBH}$ + Zbmeth*Fzbn,$_{ZBMETH}$ + Zbaob*Fzbn,$_{ZBAOB}$ + Zbnob*Fzbn,$_{ZBNOB}$ + Zbanammox*Fzbn,$_{ZBANAMMOX}$ + Zbp*Fzbn,$_{ZBP}$ + Zbpa*Fzbn,$_{ZBPA}$ + Zbam*Fzbn,$_{ZBAM}$ + Zbhm*Fzbn,$_{ZBHM}$ + X_{IN} + X_{ON} + XStru * MWNitrogen/MWStru
TKN	Total Kjeldahl Nitrogen	Total TKN is the sum of filtered and particulate TKN
TSBOD	Filtered Carbonaceous BOD	See BOD section below
TCBOD	Total Carbonaceous BOD	See BOD section below
RBCOD	Readily biodegradable COD	RBCOD is the sum the VFA-COD, the complex readily biodegradable substrate and methanol, that is: Sbsc + Sbsa + Sbsp + SbMeth
VFA	Volatile fatty acids	VFAs are calculated as the sum of acetate and propionate, that is: Sbsa + Sbsp
TN	Total N	TN is TKN and nitrate-N, that is: TKN+NO_3-N
TIN	Total inorganic N	TIN is ammonium, nitrate and struvite N, that is: NO_3 + NH_3 + (Xstru * MWNitrogen/MWStru)
Alk	Alkalinity	Alkalinity is calculated from weak acid/base chemistry – see the "*Modeling of pH in BioWin*" chapter.
pH	pH	pH is calculated from weak acid/base chemistry – see the "*Modeling of pH in BioWin*" chapter.

Project|Parameters|Other|General

Name	Default Value	Unit	Explanation

Particulate substrate COD:VSS ratio	1.60	mgCOD/mgVSS	Conversion factor between particulate substrate as measured in COD and its VSS content. The average of this value and the biomass COD:VSS ratio will lead to the typical 1.48 mgCOD/mgVSS ratio found in activated sludge.
Particulate inert COD:VSS ratio	1.60	mgCOD/mgVSS	Conversion factor between particulate inert as measured in COD and its VSS content.
Ash content of biomass (synthesis ISS)	8.00	%	Ash content of biomass. ISS generated (uptake of minerals and micronutrients) by biomass growth.
Molecular weight of other anions	35.50	mg/mmol	Assume molecular weight of "other anions" state variable.
Molecular weight of other cations	39.10	mg/mmol	Assume molecular weight of "other cations" state variable.
Mg to P mole ratio in polyphosphate	0.30	mmol Mg/mmol P	Mole ratio of magnesium to phosphorus in polyphosphate.
Cation to P mole ratio in polyphosphate	0.30	meq/mmol P	Mole ratio of magnesium to phosphorus in polyphosphate.
Ca to P mole ratio in polyphosphate	0.05	mmol Ca/mmol P	Mole ratio of calcium to phosphorus in polyphosphate.
Bubble rise velocity (anaerobic digester)	23.9	cm/s	The bubble rise velocity is used to calculate the gas holdup in anaerobic digesters. Increasing the rise velocity will decrease the gas holdup.
Bubble Sauter mean diameter (anaerobic digester)	0.35	cm	The Sauter mean diameter is the diameter of a sphere with the same volume to surface ratio as the volume to surface ratio of the total dispersion.
Anaerobic digester gas hold-up factor	1		This parameter can be used to adjust the digester mass transfer.
Tank head loss per metre of length (from flow)	0.0025	m/m	This parameter is used to calculate the power dissipation from flow.
Minimum air flow (per unit volume) without mixing	1	$m^3/(m^3 d)$	

COD and BOD in BioWin

In the BioWin simulator characterization of the carbonaceous material in municipal wastewater is in terms of the chemical oxygen demand (COD). This selection is based on a number of factors, but primarily because COD provides a consistent basis for description of the activated sludge process, and for quantifying sludge production, oxygen demand, etc. The rationale for preferring COD over other

parameters such as biochemical oxygen demand (BOD) or total organic carbon (TOC) is discussed in this section.

The simulator allows for calculation of soluble and total BOD for any input element, process unit, or stream. The user may specify the time basis for the BOD calculation (5, 7, or 20 days).

In many instances data on treatment plant operation are recorded in terms of the BOD for both the influent and effluent, usually reported as the 5-day value (BOD_5). This section discusses a number of factors regarding the application of BioWin where data are recorded on a BOD basis, or where there is a requirement to quantify effluent BOD levels for regulatory purposes.

BOD Calculations in BioWin

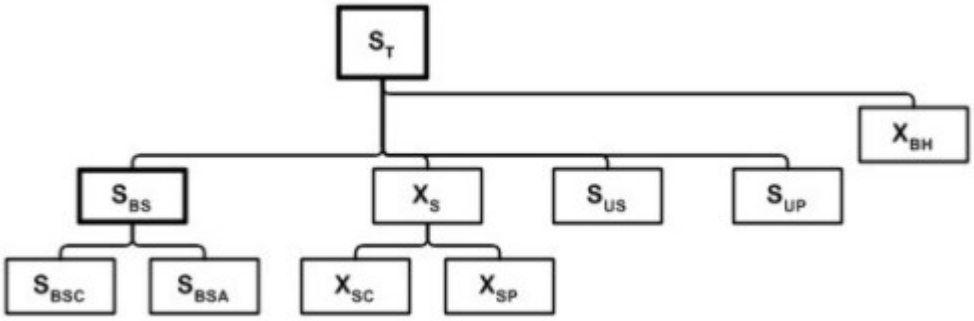
The basis for modeling organic components in BioWin is the chemical oxygen demand (COD) parameter. However, BioWin allows for calculation of soluble (filtered) and total carbonaceous biochemical oxygen demand (BOD) for any input element, process unit, or stream. The user may specify the time basis for the BOD calculation (5, 7, or 20 days).

Calculation of BOD is based on differing rates of degradation of the different components (*e.g.,* influent biodegradable material (readily and slowly biodegradable), active organism masses which exert an endogenous oxygen demand and hence a BOD, *etc*.). Essentially BioWin uses analytical equations to estimate BOD.

The objective here is to demonstrate the method used by BioWin in calculating filtered and total BOD concentrations, and show how the equations were developed. Understanding development of the equations presented below is facilitated by the figure below which illustrates the division of a wastewater stream into the different components.

Definition of Terms	
B	Endogenous decay rate constant ($\cong 0.24\ d^{-1}$)
BOD_E	Biochemical oxygen demand due to endogenous respiration
BOD_S	Soluble component of biochemical oxygen demand
BOD_{XSP}	Particulate slowly biodegradable component of biochemical oxygen demand
BOD_{XBH}	Active biomass component of biochemical oxygen demand
BOD_T	Total biochemical oxygen demand
f	Fraction of active mass remaining as endogenous residue ($\cong 0.20$)
f_{BS}	Fraction of total influent COD which is readily biodegradable
f_{UP}	Fraction of total influent COD which is unbiodegradable particulate
f_{US}	Fraction of total influent COD which is unbiodegradable soluble
f_{XBH}	Fraction of total influent COD which is active organisms
f_{XSP}	Fraction of slowly degradable COD which is particulate
K	First order rate constant for X_{SP} degradation ($\cong 0.40\ d^{-1}$)
$(MO_2)_G$	Mass of oxygen utilized for growth on soluble substrate
$(MO_2)_E$	Mass of oxygen utilized for endogenous metabolism
OUR	Oxygen utilization rate (mg/L/day)

OUR_E	Oxygen utilization rate due to endogenous metabolism
OUR_G	Oxygen utilization rate for substrate utilization (growth)
S_{BS}	Soluble readily biodegradable COD concentration (mg COD/L)
S_{BSA}	Soluble readily biodegradable volatile fatty acid COD concentration (mg COD/L)
S_{BSP}	Soluble readily biodegradable propionate COD concentration (mg COD/L)
S_{BMETH}	Soluble readily biodegradable methanol COD concentration (mg COD/L)
S_{BSC}	Soluble readily biodegradable complex COD concentration (mg COD/L)
S_S	Soluble (filtered) biodegradable COD concentration (mg COD/L)
S_{UP}	Particulate unbiodegradable COD concentration (mg COD/L)
S_{US}	Soluble unbiodegradable COD concentration (mg COD/L)
S_T	Total COD concentration (mg COD/L)
T	Time (d)
X_{BH}	Active organism concentration (*i.e.* the sum of seven organism concentrations) (mg COD/L)
X_{BH0}	Active organism concentration at time zero (mg COD/L)
X_S	Slowly biodegradable COD concentration (mg COD/L)
X_{SC}	Slowly biodegradable colloidal COD concentration (mg COD/L)
X_{SP}	Slowly biodegradable particulate COD concentration (mg COD/L)
X_{SP0}	Slowly biodegradable particulate COD concentration at time zero (mg COD/L)
Y	Yield of active organisms (≅ 0.666 mg cell COD mg COD^{-1})



Division of municipal wastewater COD into constituent fractions

Basis for BOD Calculations

The approach for calculating BOD is to distinguish three components, and to calculate the BOD contribution for each component independently. These components are as follows:

1. BOD associated with utilization of soluble COD (both readily biodegradable and colloidal slowly biodegradable) and the biomass generated from this utilization;

2. BOD associated with utilization of slowly biodegradable particulate COD and the biomass generated from this utilization; and
3. BOD exerted by active biomass present in the sample. That is, biomass initially present; not biomass generated through utilization referred to in 1 and 2.

The simulator provides values for S_{BSC}, S_{BSA}, X_{SC}, X_{SP} and X_{BH} in any stream or process unit. When calculating BOD concentrations for input streams, the influent fractions provided by the user (or the default values) are used to calculate the appropriate concentrations.

BOD Associated with Soluble Biodegradable COD (S_S):

For the purpose of BOD calculations, the "soluble" biodegradable COD (S_S) can be considered to be the sum of the readily biodegradable COD ($S_{BSC} + S_{BSA} + S_{BSP} + S_{BMETH}$) and the biodegradable colloidal COD (X_{SC}), *i.e.,*

$$S_S = S_{BSC} + S_{BSA} + S_{BSP} + S_{BMETH} + X_{SC} \quad (1)$$

$$S_S = S_{BSC} + S_{BSA} + S_{BSP} + S_{BMETH} + X_{SC} \quad (1)$$

The oxidation of this material occurs rapidly, and for the purpose of these calculations is assumed to occur instantaneously (*i.e.,* at t = 0). Assuming that the soluble biodegradable COD is oxidized instantly to form new cells:

$$S_S \rightarrow X_{BH0} = Y \cdot S_S \quad (\text{at time } t = 0) \quad (2)$$

then the mass of oxygen consumed for growth of these organisms $(MO_2)_G$ is:

$$(MO)_G = (1 - Y) \cdot S_S \quad (3)$$

Assuming that the rate of change in organism concentration due to endogenous decay is first order with respect to the active organism concentration, an expression for active organism concentration can be obtained, *i.e.*,

$$\frac{dX_{BH}}{dt} = -b \cdot X_{BH} \quad (4)$$

Therefore,

$$X_{BH} = X_{BH0} \cdot e^{-b\,t} \quad (5)$$

The BOD due to endogenous metabolism by the organisms generated from growth on S_S can then be calculated using either of the methods described below.

Method 1:

From Eq.2 and Eq.5 , the endogenous mass loss at time t can be calculated:

$$\begin{aligned} \Delta X_{BH} &= X_{BH0} - X_{BH} \\ &= X_{BH0} \cdot (1 - e^{-b\,t}) \\ &= Y \cdot S_S \cdot (1 - e^{-b\,t}) \end{aligned} \quad (6)$$

and the mass of oxygen consumed for endogenous metabolism:

$$
\begin{aligned}
(MO)_E &= (1-f)\cdot \Delta X_{BH} \\
&= (1-f)\cdot Y \cdot S_S \cdot (1-e^{-b\,t})
\end{aligned}
\tag{7}
$$

This is equal to the BOD due to endogenous decay, *i.e.,*

$$
\begin{aligned}
BOD_E &= (1-f)\cdot X_{BH0}\cdot (1-e^{-b\,t}) \\
&= (1-f)\cdot Y \cdot S_S \cdot (1-e^{-b\,t})
\end{aligned}
\tag{8}
$$

The soluble BOD component can now be calculated by summing Eq. 3 and Eq. 8, *i.e.,*

$$
\begin{aligned}
BOD_S &= (MO)_G + (MO)_E \\
&= (1-Y)\cdot S_S + (1-f)\cdot Y \cdot S_S \cdot (1-e^{-b\,t})
\end{aligned}
\tag{9}
$$

Method 2:

The OUR due to endogenous metabolism can also be related to the rate of change in active organism concentration (taking into account that a fraction of the active mass becomes endogenous residue), *i.e.,*

$$
\begin{aligned}
OUR_E &= -(1-f)\cdot \frac{dX_{BH}}{dt} \\
&= b\cdot (1-f)\cdot X_{BH} \\
&= b\cdot (1-f)\cdot X_{BH0}\cdot e^{-b\,t}
\end{aligned}
\tag{10}
$$

The BOD due to endogenous decay is then the cumulative mass of oxygen used over time, *i.e.,*

$$
\begin{aligned}
BOD_E &= \int_0^t OUR_E\, dt \\
&= b\cdot (1-f)\cdot X_{BH0}\int_0^t e^{-b\,t} dt \\
&= b\cdot (1-f)\cdot X_{BH0}\cdot \left(-\frac{1}{b}e^{-b\,t}\right) + C \\
&= -(1-f)\cdot X_{BH0}\cdot e^{-b\,t} + C
\end{aligned}
\tag{11}
$$

and since at t = 0, $BOD_E = 0$,

$$C = (1+f)\cdot X_{BH0}$$

Substituting the expression obtained for X_{BHO} (Eq. 2) into the above gives an expression for the BOD due to endogenous decay:

$$
\begin{aligned}
BOD_E &= -(1-f)\cdot X_{BH0}\cdot e^{-b\,t} + (1-f)\cdot X_{BH0} \\
&= (1-f)\cdot X_{BH0}\cdot (1-e^{-b\,t}) \\
&= (1-f)\cdot Y \cdot S_S \cdot (1-e^{-b\,t})
\end{aligned}
\tag{12}
$$

This is the same expression obtained using Method 1 (Eq. 8). The total soluble BOD portion is given by Eq. 9.

BOD Associated with Slowly Biodegradable Particulate COD (X_{SP}):

The biochemical oxygen demand related to the slowly biodegradable particulate material can be calculated from the cumulative oxygen consumption for growth on X_{SP} and endogenous respiration exerted by organisms from this growth. The OUR is the sum of two components:

$$\begin{aligned} OUR &= OUR_G + OUR_E \\ &= (1 - Y) \cdot \left(- \frac{dX_{SP}}{dt} \right) + (1 - f) \cdot \left(- \frac{dX_{BH}}{dt} \right)_E \\ &= (1 - Y) \cdot \left(- \frac{dX_{SP}}{dt} \right) + (1 - f) \cdot \left(bX_{BH} \right) \end{aligned} \tag{13}$$

The rate of change in slowly biodegradable particulate substrate concentration is assumed to be first order with respect to substrate concentration, *i.e.*,

$$\frac{dX_{SP}}{dt} = -k \cdot X_{SP}$$

Therefore,

$$X_{SP} = X_{SP0} \cdot e^{-kt} \tag{14}$$

The active organism concentration in Eq. 11 is obtained by integrating an expression for the rate of change of X_{BH}. This rate is a consequence of an increase due to growth on X_{SP}, minus a decrease due to endogenous metabolism, *i.e.*,

$$\frac{dX_{BH}}{dt} = Y \cdot \left(- \frac{dX_{SP}}{dt} \right) - b \cdot X_{BH} \tag{15}$$

Substituting Eq. 14 into the above results in a linear differential equation for X_{BH} as a function of time, *i.e.,*

$$\begin{aligned} \frac{dX_{BH}}{dt} &= - Y \cdot \frac{dX_{SP}}{dt} - b \cdot X_{BH} \\ &= -Y \frac{d}{dt} \left(X_{SP0} \cdot e^{-kt} \right) - b \cdot X_{BH} \\ &= kYX_{SP0} \cdot e^{-kt} - b \cdot X_{BH} \end{aligned} \tag{16}$$

Or equivalently,

$$\frac{dX_{BH}}{dt} + b \cdot X_{BH} = kYX_{SP0} \cdot e^{-kt} \tag{17}$$

An integration factor of e^{bt} can be used to solve the differential equation. Multiplying both sides of the equation by e^{bt} results in a simplified expression:

$$\begin{aligned} e^{bt} \cdot \frac{dX_{BH}}{dt} + b \cdot e^{bt} \cdot X_{BH} &= e^{bt} \cdot kYX_{SP0} \cdot e^{-kt} \\ \frac{d}{dt} (e^{bt} \cdot X_{BH}) &= kYX_{SP0} \cdot e^{(b-k)t} \end{aligned} \tag{18}$$

Integrating both sides of Eq. 18:

$$(e^{bt} \cdot X_{BH}) = \int kYX_{SP0} \cdot e^{(b-k)t}\, dt$$

$$(e^{bt} \cdot X_{BH}) = \frac{kYX_{SP0}}{(b-k)} \cdot e^{(b-k)t} + C \tag{19}$$

$$X_{BH} = \left(\frac{kYX_{SP0}}{b-k}\right) \cdot e^{-kt} + Ce^{-bt}$$

At t = 0 $X_{BH} = X_{BHO}$, therefore solving for **C** gives:

$$X_{BH} = \left(\frac{kYX_{SP0}}{b-k}\right) \cdot e^{-kt} + \left(X_{BH0} - \frac{kYX_{SP0}}{b-k}\right) \cdot e^{-bt} \tag{20}$$

Returning to Eq. 13 for the oxygen utilization rate and substituting for X_{BH}:

$$\begin{aligned} OUR &= (1-Y) \cdot \left(-\frac{dX_{SP}}{dt}\right) + (1-f) \cdot (bX_{BH}) \\ &= (1-Y) \cdot kX_{SP} + (1-f) \cdot bX_{BH} \\ &= (1-Y) \cdot kX_{SP0} \cdot e^{-kt} + (1-f) \cdot b \cdot \left[\left(\frac{kYX_{SP0}}{b-k}\right) \cdot e^{-kt} + \left(X_{BH0} - \frac{kYX_{SP0}}{b-k}\right) \cdot e^{-bt}\right] \end{aligned} \tag{21}$$

Integrating Eq. 21 gives an expression for the BOD of the slowly biodegradable material as a function of time, *i.e.,*

$$\begin{aligned} BOD_{XSP} &= \int_0^t OUR\, dt \\ &= \int_0^t (1-Y) \cdot kX_{SP0} \cdot e^{-kt} + (1-f) \cdot b \cdot \left[\left(\frac{kYX_{SP0}}{b-k}\right) \cdot e^{-kt} + \left(X_{BH0} - \frac{kYX_{SP0}}{b-k}\right) \cdot e^{-bt}\right] dt \\ &= -(1-Y) \cdot X_{SP0} \cdot e^{-kt} - \frac{(1-f) \cdot bYX_{SP0}}{(b-k)} \cdot e^{-kt} - (1-f) \cdot \left(X_{BH0} - \frac{kYX_{SP0}}{b-k}\right) \cdot e^{-bt} \\ &\quad + (1-Y) \cdot X_{SP0} + \frac{(1-f) \cdot bYX_{SP0}}{(b-k)} + (1-f) \cdot \left(X_{BH0} - \frac{kYX_{SP0}}{b-k}\right) \end{aligned}$$

Assuming $X_{BHO} \approx 0$ and re-arranging the above expression,

$$BOD_{XSP} = X_{SP0} \cdot \left[\left((1-Y) + \frac{(1-f) \cdot bY}{(b-k)}\right) \cdot \left(1 - e^{-kt}\right) - \left(\frac{(1-f) \cdot Yk}{(b-k)}\right) \cdot \right. \tag{22}$$

The total BOD (if no biomass is present) is then the sum of BOD_S and BOD_{XSP},. (Eqs. 9 and 22), *i.e.,*

$$\begin{aligned} BOD_T &= X_{SP0} \cdot \left[\left((1-Y) + \frac{(1-f) \cdot bY}{(b-k)}\right) \cdot \left(1 - e^{-kt}\right) - \left(\frac{(1-f) \cdot Yk}{(b-k)}\right) \cdot \left(1 - e^{-bt}\right)\right] \\ &\quad + (1-Y) \cdot S_S + (1-f) \cdot Y \cdot S_S \cdot (1 - e^{-bt}) \end{aligned} \tag{23}$$

BOD Associated with Active Biomass

In cases where there are active organisms present, the BOD exerted by the organisms can be calculated from the rate of change in active organisms due to endogenous metabolism, *i.e.,*

$$BOD_{XBH} = (1 - f) \cdot X_{BH} \cdot (1 - e^{-bt}) \quad (24)$$

This must then be added to the soluble and slowly degradable BODs to obtain the total BOD (*i.e.*, sum Eqs. 23, and 24).

Example

This example demonstrates the BOD calculation procedure for an influent wastewater stream with the following characteristics:

S_T	500 mg COD/L
f_{US}	0.10
f_{UP}	0.08
f_{BS}	0.20
f_{XBH}	0.00
f_{XSP}	0.75

The influent "soluble" and slowly biodegradable concentrations are calculated using the influent fractions:

$$S_{BS} = f_{BS}\, S_T$$

$$= (0.20)(500)$$

$$= 100 \text{ mg COD / L}$$

$$X_S = (1 - f_{US} - f_{UP} - f_{BS} - f_{XBH})\, S_T$$

$$= (1 - 0.10 - 0.08 - 0.20 - 0.0)(500)$$

$$= 310 \text{ mg COD / L}$$

$$X_{SC} = (1 - f_{XSP})\, X_S$$

$$= (1 - 0.75)(310)$$

$$= 77.5 \text{ mg COD / L}$$

$$X_{SP} = (f_{XSP})\, X_S$$

$$= (0.75)(310)$$

$$= 232.5 \text{ mg COD / L}$$

$$S_S = S_{BS} + X_{SC}$$
$$= 100 + 77.5$$
$$= 177.5 \text{ mg COD / L}$$

The soluble BOD_5 can then be calculated using Eq.9 and the parameters listed below:

f	0.20
Y	0.666 mg COD/mg COD
b	0.24 d^{-1}
k	0.40 d^{-1}

Note: The parameter values for *f* and b are for endogenous respiration, and should not be confused with the corresponding terms used in the death-regeneration approach for modeling biomass decay.

$$BOD_S = (1 - Y) \cdot S_S + (1 - f) \cdot Y \cdot S_S \cdot (1 - e^{-b\,t})$$

$$= (1 - 0.666)(177.5) + (1 - 0.20)(0.666)(177.5)(1 - e^{-(0.24)(5)})$$

$$= 125.4 \text{ mg BOD / L}$$

Similarly, the slowly degradable BOD_5 can be calculated using Eq. 22.

$$BOD_{XSP} = X_{SP0} \cdot \left[\left((1 - Y) + \frac{(1 - f) \cdot b\,Y}{(b - k)} \right) \cdot \left(1 - e^{-k\,t}\right) - \left(\frac{(1 - f) \cdot Yk}{(b - k)} \right) \cdot \left(1 - e^{-b\,t}\right) \right]$$

$$= (232.5) \left[\begin{array}{l} \left((1 - 0.666) + \dfrac{(1 - 0.20)(0.24)(0.666)}{(0.24 - 0.40)} \right) \cdot (1 - e^{-(0.40)(5)}) \\ \qquad - \left(\dfrac{(1 - 0.20)(0.666)(0.40)}{(0.24 - 0.40)} \right) \cdot (1 - e^{-(0.24)(5)}) \end{array} \right]$$

$$= 122.9 \text{ mg BOD / L}$$

In this example the active organism fraction is zero, therefore the total BOD_5 is the sum of the BOD_S and the BOD_{XSP}, *i.e.,*

$$BOD_T = BOD_S + BOD_{XSP}$$
$$= 125.4 + 122.9$$
$$= 248.3 \text{ mg BOD / L}$$

Effluent BOD

Knowledge of the effluent BOD is useful for quantifying the impact of plant discharges on receiving water bodies, and often is of interest in terms of regulatory requirements and consent limits. Currently BioWin provides information on a number of biodegradable components in any plant output stream (and internal streams). Therefore it is a simple step to quantify both the soluble and total (soluble + particulate) BOD. In BioWin this calculation is based on differing rates of degradation of the different components [e.g. undegraded influent biodegradable material (readily and slowly biodegradable), active organism masses which exert an

endogenous oxygen demand and hence a BOD, etc.]. Essentially BioWin simulates a BOD test for the specified number of days and wastewater composition. This should allow for more accurate estimation of BOD than is provided with models based on the BOD parameter.

COD *versus* BOD as a Modeling Parameter

The advantage of selecting COD as the parameter for quantifying the "strength" of organic material in the influent, as opposed to BOD or TOC, is that it provides a consistent basis for description of the activated sludge process. Marais and Dold (1985) outlined the rationale which makes COD the appropriate parameter. It is worth reviewing this rationale briefly as selection of the COD parameter is fundamental to the application of the models. Briefly, the suitability of COD is established by considering utilization of organic substrate. In the process of metabolism the organic substrate serves two functions for the organisms as shown schematically in the arrow diagram below:

1. A portion of the organic material is oxidized to CO_2 and water, providing energy for maintaining the homeostatic balance for existing cell mass (osmotic pressure, ionic balance, membrane potential, etc.) and for (2) below. The energy is provided by transferring electrons from the organic substrate through the electron transport chain to the terminal electron acceptor (oxygen in the case of an aerobic system, or nitrate under anoxic conditions). Under the substrate limited conditions usually encountered in activated sludge systems the organisms utilize a relatively fixed fraction of the energy available from oxidation for the energy consuming processes.
2. The remaining portion of the organic material is converted into new heterotrophic cell mass, utilizing energy available from process (1).

Regarding the relative proportions allocated between (1) and (2), this is quantified by the ratio:

$$Y_H = \frac{\text{cell mass formed}}{\text{substrate utilized}} \tag{25}$$

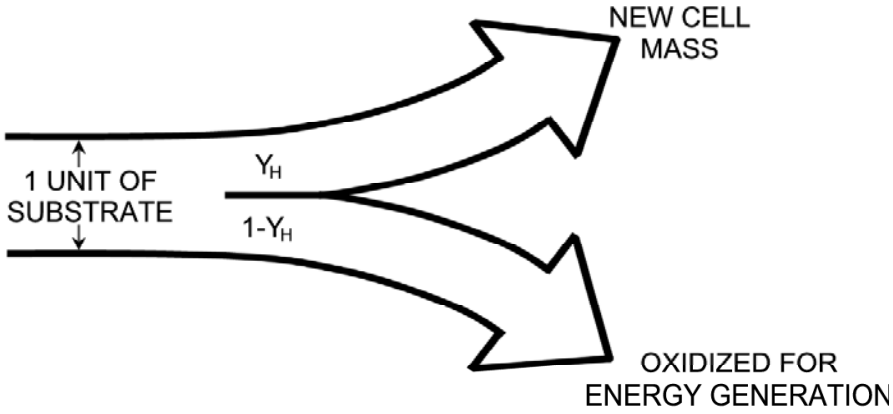
termed the specific yield. From theoretical considerations, under balanced growth conditions the specific yield, in terms of the mass of organisms produced *per electron available for transfer* in the biodegradable organic substrate, should be near constant. This was confirmed by Payne (1970) who reviewed a large number of experimental results for both pure and mixed cultures of heterotrophic organisms. Therefore, if the electron donating potential of the organic substrate in the activated sludge system influent (consisting of a broad spectrum of compounds) is measured, it is possible to quantify sludge production [from Y_H] and oxygen demand [from (1 - Y_H)]. This observation provides the rationale for selecting the COD parameter.

The electron donating capacity of organic material is measured in the COD test. In the test each mole of oxygen (O_2) accepts four electron equivalents (e^- eq); therefore the COD is a direct measure of the electron donating potential. The link between electron equivalents (COD) of the substrate, the near constant yield of organism mass per unit substrate COD, and the corresponding fixed oxygen requirement per unit substrate COD metabolized, make the COD a fundamental parameter in the analysis of activated sludge behavior.

Regarding the COD of the waste sludge, this may be measured directly by the COD test or may be estimated from the VSS measurement. Because cell mass has an approximately constant composition and is made up of an essentially fixed number of electron equivalents (e- eq) per unit mass, the COD/VSS ratio is near constant (approximately 1.48 mgCOD/mgVSS).

A factor which adds impetus to the selection is that mass balances can be made in terms of COD. As electrons cannot be created or destroyed, in a biosystem operated at steady state the mass of COD entering with the influent per unit time must equal the sum of (1) the mass of COD leaving in the effluent, (2) the COD of the wasted sludge, and (3) the oxygen (or oxygen equivalent) consumed in the utilization of the organic material (from the oxygen utilization rate measurement); that is, a mass balance is possible.

Characteristics of the COD as a measure of the "strength" of a wastewater are not realized by either the BOD or the TOC parameters. The BOD measures only that portion of the e^- eq in the substrate utilized for energy generation and excludes the portion of the substrate e^- eq transformed into new cell mass. Therefore BOD cannot be used as the basis for a mass balance. The TOC is deficient in that the ratio of carbon/e^- eq differs between organic compounds and therefore TOC is an inappropriate parameter when dealing with the mixed substrate influents to wastewater treatment plants.



Schematic representation of the utilization of substrate by heterotrophic organisms showing the division between substrate oxidized for energy generation and growth of new cell mass.

Note: The user must specify influent concentrations in terms of COD; however, the simulator can estimate the soluble and total BOD for any input element, process unit, or stream. The user may specify the time basis for the BOD calculation (5, 7, or 20 days).

Settling and Solid / Liquid Separation Models

Types of Models

In BioWin there are three basic types of models used in solid / liquid separation elements:

1. **Point (i.e. dimensionless)**
2. **Ideal (similar to point models, but with volume)**
3. **Flux based**

This chapter outlines the theory behind the different model types and gives examples of BioWin elements that they are used in.

Point Separation Models

Point separation models perform a simple mass balance calculation to split the incoming solids into two streams. The percentage removal specified by the user is based on a mass basis.

For example, if a user specifies that a point separator has a 95% solids capture, the point separator reports 95% of the incoming total suspended solids mass to the "thickened" stream (e.g. the underflow of a point settling tank), and the remaining 5% reports to the "clarified" stream (e.g. the overflow of a point settling tank).

Examples of BioWin elements that use point separation models include **Dewatering Units** and **Point Secondary Settlers**.

Ideal Separation Models

Ideal solid / liquid separation models are very similar to point separation models except for the fact that they have volume. The total volume of BioWin units employing ideal solid / liquid separation models is divided into two sub-volumes (a "thickened" or "sludge" volume and a "clarified" or "liquid" volume – the relative volume proportions are specified by the user).

At steady state conditions, the mass coming out of the sludge volume zone will be the same as the mass entering it, and specifying the flow split (e.g. the underflow rate

for an ideal secondary settler) and the percent solids removal will completely define the mass balance around the unit.

Under dynamic loading, the mass coming out of the sludge volume may not be the same as that coming in; however it may not fluctuate as much as that coming in because the sludge zone volume has an attenuating effect. Consider the following mass balance equation on TSS for the sludge zone, assuming a 95% solids removal:

$$\text{Accumulation} = \text{Mass In} - \text{Mass Out}$$

$$\frac{\partial C}{\partial t} \cdot V = 0.95 \cdot Q_{IN} \cdot C_{IN} - Q_{OUT} \cdot C_{OUT} \quad (1)$$

where C = TSS concentration (kg/m^3), V = sludge zone volume (m^3), Q = Flow rate (m^3/d).

For the steady state case, the change in concentration with respect to time is zero, so the left hand side of the above equation goes to zero. Then the concentration out the bottom (C_{OUT}) may be obtained algebraically. But for the dynamic case, this term may not necessarily be zero - it could be positive or negative depending on what is happening in the sludge zone, and in this case, the term involving the volume of the thickened sludge zone does not drop out to zero. Therefore in a dynamic loading case, the volume of the sludge zone will affect the concentration coming out the bottom. How much of an impact it has depends on the volume, the variability of the incoming load, etc.

Examples of BioWin elements that use ideal separation models include **Ideal Secondary Settlers**, **Ideal and Activated Primary Settling Tanks**and **Grit Removal Tanks**.

Note: Grit removal elements are special cases that act only on inorganic suspended solids (ISS).

Flux Based Models

The general flux theory approach models solids settling in one dimension. In the one-dimensional approach solids and liquid movement in the vertical direction are assumed to be dominant and horizontal movement is ignored. The settling tank is divided into a number of layers in the vertical direction and a numerical technique is used to solve the mass balance equations in the vertical direction. The solution to the mass balance equations provides the solids concentration profile in the settling tank, and the solids concentration in the effluent and underflow.

When considering a one-dimensional model of a secondary settling tank, one must consider the behavior in three separate zones:

1. The zone above the feed layer.
2. The feed layer.
3. The zone below the feed layer.

This is necessary because the mass balance equations that describe a given layer change from zone to zone. Above the feed layer, the bulk fluid movement (i.e. surface overflow rate) is upwards; therefore any solids transport associated with the

bulk fluid movement is upwards as well. Below the feed layer, the bulk fluid movement (i.e. underflow rate) is downwards, so solids transport associated with the bulk fluid movement also is downwards. At the feed layer, the solids mass loading of the feed must be considered, and there is both upwards and downwards bulk fluid movement. [It is assumed that the feed flow is the sum of the overflow and the underflow, and the "flow split" occurs in this layer]. Also, the top layer in the first zone (i.e. overflow) and the bottom layer in the third zone (i.e. underflow) require special consideration. The top layer in the first zone is unique as there is no solids flux into it. The bottom layer in the third zone is unique as there only is bulk flux out of it. Another requirement of one-dimensional secondary settling tank models is a quantitative relationship between solids concentration and settling velocity.

BioWin offers two types of flux theory based models:

1. **Modified Vesilind**
2. **Double Exponential**

Common Feature: Locate a Specified Suspended Solids Concentration

A very powerful feature of both settling models is that a user can specify a suspended solids concentration and the model will locate and plot the height of that suspended solids concentration within the settler profile.

For example, if a user has measured the concentration of the sludge blanket in a secondary settling tank to be 2,500 mg/L the model can locate the height of this concentration within its profile, and plot it with time. This gives users a tool to track the approximate location of the sludge blanket.

Modified Vesilind Secondary Settler Model

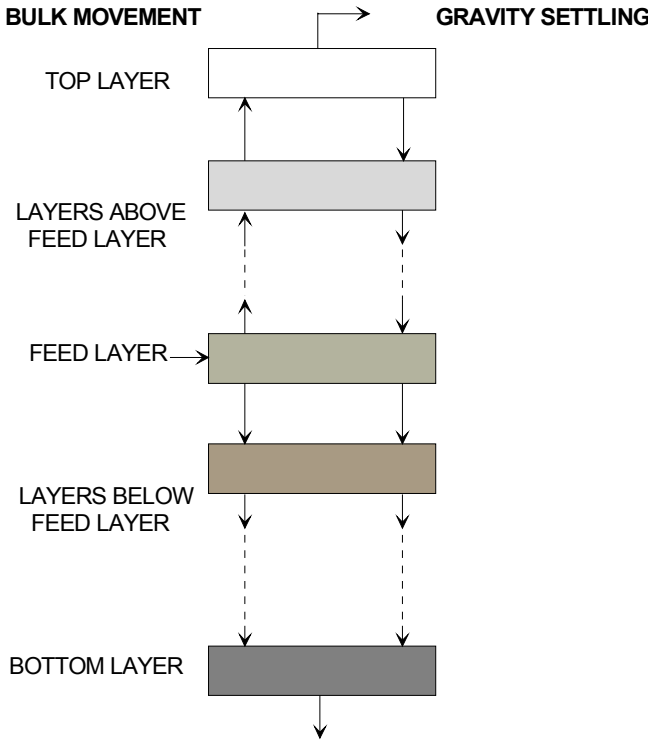
The approach used in this model is to divide the settling tank into a number of layers (minimum of 5), with a solids concentration in each layer, resulting in an essentially one-dimensional model for sludge settling.

The model is based essentially on standard solids flux analysis. That is, the mass flux of solids out of each layer is assumed to be the sum of the gravity settling flux and the flux due to bulk movement (refer to the diagram below).

Note: Fewer layers may overestimate the mass of solids on the base (i.e. bottom layer) of the settling tank. The underflow **TSS** concentration essentially is determined by the thickening factor **$[(Q_I + Q_R) / Q_R]$**, and this will be the concentration in the bottom layer. The height of the bottom layer (settler depth divided by the number of layers) largely determines the mass of sludge within an underloaded settling tank, so the layer depth should correspond to the depth of sludge on the base of the settling tank. For example, to simulate the case of say a 1 foot deep sludge layer in a 15 foot deep settling tank, set the number of layers to 15.

For steady state calculations, BioWin uses a non-linear equation solver to determine the solids concentration in each layer of the settler. A set of equations is obtained by generating mass balance expressions for the solids in each settler layer. Solids entering the settling tank are assumed to be distributed instantaneously across the feed layer.

Solving the set of equations is not a trivial mathematical problem due to the exponential nature of the sludge settling velocity expression (described in more detail below). Simulation problems can arise in situations where the solids loading rate to the settler is high (and the settler would probably fail in practice). These problems are due to the fact that, under certain circumstances, the set of mass balance equations (one for each layer) can have multiple solutions. If this is encountered during the iterative solution procedure BioWin may not be successful in moving away from a non-feasible solution to the correct solution.



Representation of layered approach used in the Modified Vesilind settler model.

Bulk flux terms are given by the product of the up or downflow velocity and layer concentration, i.e.:

$$J_{BULK} = \frac{Q_I}{A} \cdot X_i \text{ (above feed layer)}$$
$$J_{BULK} = \frac{Q_R}{A} \cdot X_i \text{ (below feed layer)} \quad (2)$$

where Q_I = plant influent flow, Q_R = settling tank uderflow rate, A = settling tank cross-sectional area, and X_i = suspended solids concentration in layer *i*.

Gravity flux terms are given by the product of settling velocity and layer concentration, i.e.:

$$J_{S,i} = V_{S,i} \cdot X_i \quad (3)$$

where $V_{S,i}$ = settling velocity in layer *i* and X_i = suspended solids concentration in layer *i*.

Sludge Settling Velocity

The development of a model for the settling tank requires an expression for the settling velocity of the sludge. In order to obtain this expression it is necessary to make a number of assumptions. Firstly, zone settling (or "hindered" settling) is assumed to be the main type of settling occurring in the settling tank. That is, there is a distinct interface between settling particles and clarified liquid (i.e. the inter-particle forces are sufficient for all particles in a given cross section to settle at the same rate regardless of particle size). This means that the solids flux through the settling tank, which is given by the product of the solids concentration and the settling velocity, varies with depth in the settling tank.

Sludge settling velocity is modeled according to the Vesilind equation for hindered settling. The settling velocity in a given layer is given by:

$$V_{S,i} = V_0 \, e^{-KX_i} \tag{4}$$

where V_0 = maximum settling velocity (m/d), K = settling parameter (m3/kgTSS), and X_i = total suspended solids (TSS) concentration (kgTSS/m3) in layer *i*.

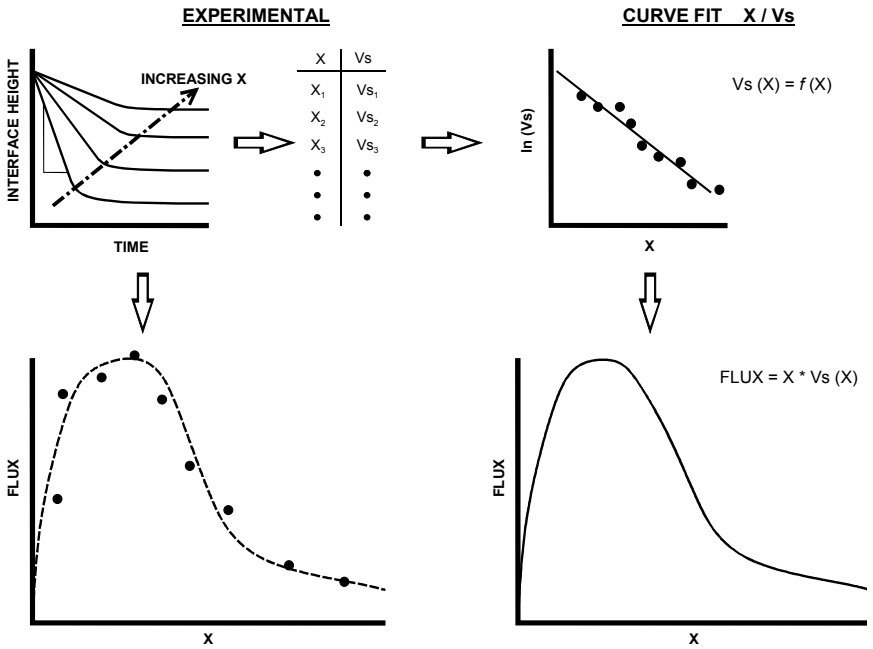
The above equation can be fit to experimental data by regression to yield values for the settling parameters V_0 and K. A number of batch settling tests should be conducted to determine the zone settling velocity (V_S) over a range of suspended solids (X) concentrations. Referring to the diagram below, the zone settling velocity for a given solids concentration is determined from the slope of the first straight line portion of the interface height versus time plot as shown. By taking the natural logarithm of both sides of the above equation, the following "linearized" expression is obtained:

$$\ln V_S = \ln V_0 - K \cdot X \tag{5}$$

A semi-log plot of $\ln(V_S)$ versus X can then be used to estimate V_0 and K. The parameter K is given by the slope and V_0 by the intercept of the line of best fit through the experimental data.

A flux curve can then be generated as a continuous function using the Vesilind expression for settling velocity and the parameters derived from the experimental data (as shown). The expression for the mass flux of solids can then be used in the mass balance expressions for each settler layer (the total flux being the sum of the gravity settling flux, as described by the Vesilind equation, and the flux due to the bulk movement of the liquid).

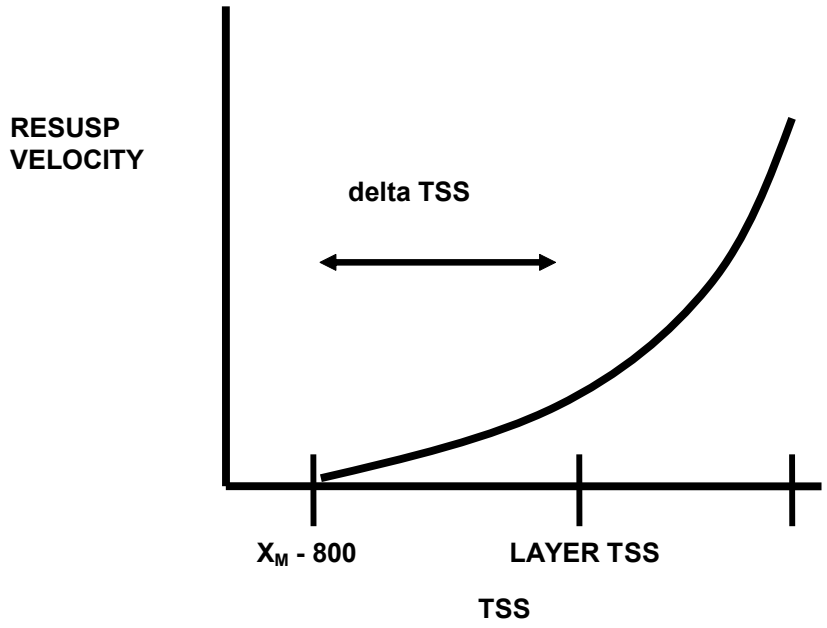
Due to the extensive amount of experimental testing required to determine Vesilind model parameters, a number of correlations have been developed in order to estimate Vesilind model parameters from settleability measures such as SVI, DSVI, etc. Extreme care should be taken in using these correlations, as their applicability may be limited [Bye & Dold (1998), Bye & Dold (1999)].



Procedure used to obtain Vesilind settling velocity model parameters.

Solids Resuspension and Compactability

The Modified Vesilind settling model uses a maximum solids compactability in an attempt to mimic what takes place in the bottom layer of a settling tank. As the solids concentration approaches the maximum solids compactability resuspension of solid particles occurs. The rate of solids resuspension is assumed to be proportional to the square of the difference between the solids concentration in a given layer and the maximum solids compactability. The resuspension term essentially switches on as the solids concentration in a layer approaches the maximum allowable. Increasing the solids compactability will reduce the rate of resuspension, resulting in fewer solids in the settling tank effluent. The following figure illustrates the concept of solids resuspension.



Representation of resuspension behavior in bottom layer

Resuspension begins at solids concentration values slightly less than the maximum compactability X_M. Referring to the above diagram, the offset is 800 mg/L less than the maximum compactability. Once the solids concentration in a layer gets above this offset concentration, resuspension begins.

The resuspension velocity is proportional to the square of the difference between the layer solids concentration and the offset solids concentration (i.e. delta TSS in the above diagram).

The resuspension velocity is then multiplied by the layer solids concentration to calculate the resuspension flux term. This is done because the modified Vesilind settler model is flux-based and tracks the various flux terms (e.g. Vesilind, bulk, resuspension) that enter and leave layers.

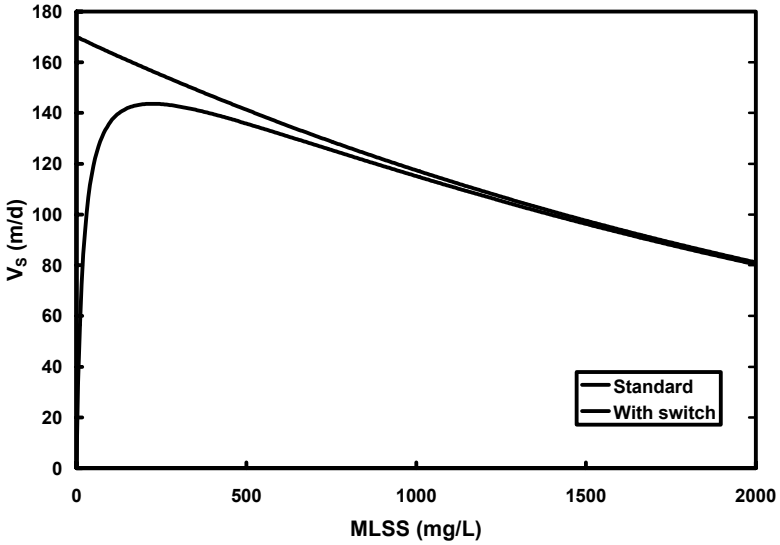
Settling Velocity Switching Function

A criticism of Eq. 4 is that it over-predicts settling velocities at low concentrations. BioWin employs a switching function to switch off the sludge settling velocity expression at very low solids concentrations. This provides a simplified method of modeling the poorly flocculating solids in the upper layers of the settling tank. As the solids concentration approaches the settling velocity TSS switch the settling velocity approaches zero, and solids are carried upwards by the overflow stream (resulting in solids in the settling tank effluent).

The settling velocity equation used by BioWin is:

$$V_{S,i} = V_0 \; e^{-KX_i} \cdot \left(\frac{X_i}{K_S + X_i} \right) \tag{6}$$

where K_S is the settling velocity switch, and other variables are the same as in Eq. 4. The switching function effect for a given combination of settling velocity parameters is shown in the figure below:



The settling velocity switch reduces the settling velocity at low concentration levels

You can increase the suspended solids concentration at which settling velocity is decreased by increasing K_S. Varying the K_S parameter is a possible way to vary the predicted effluent suspended solids.

Avoiding Modified Vesilind Settler Model Problems

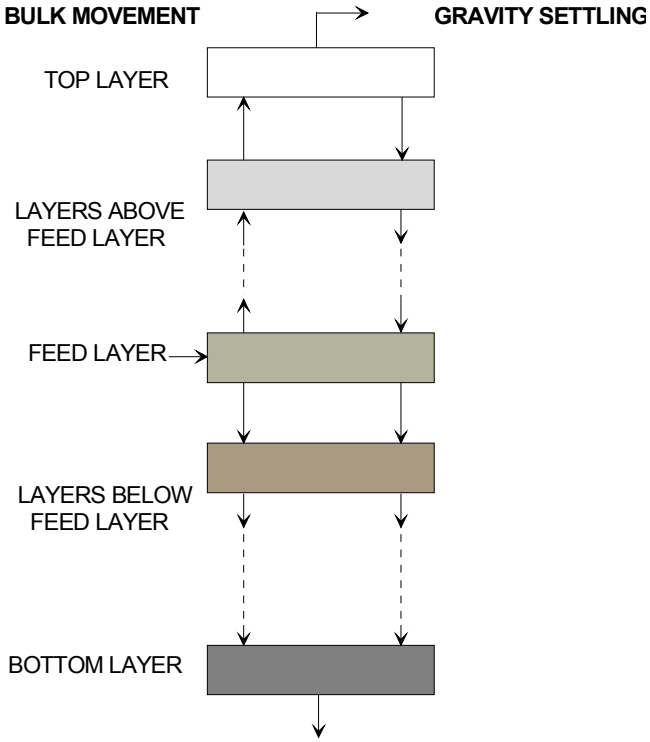
BioWin's steady state solution algorithm has been improved significantly, and the Modified Vesilind settling tank model is more robust as a result. Occasionally users may encounter situations where the solver requires many iterations to move to a solution. Often these problems can be addressed by increasing the maximum solids compactability.

Other hints for handling settler problems are given below:

1. It may be desirable to leave the settler profile window open during a simulation; this will allow the user to observe the movement of the sludge blanket throughout the simulation, and indicates when the settler is becoming overloaded.
2. If the steady state solver cannot find a solution, then subsequent dynamic simulations should allow time for the settler to approach a solution (i.e. run the simulation over several sludge ages).
3. In order to obtain an accurate solution under dynamic conditions it may be necessary to run the simulation over several sludge ages, until the starting point for the simulation reflects the actual conditions of the system at the start of the time period being simulated (i.e. solids concentration in the bioreactor, position of sludge blanket, etc. are reasonably close to initial conditions of system).

Double Exponential Settler Model

The Double Exponential settler model overcomes many of the difficulties that may be encountered with unrestricted solids flux models (e.g. the Modified Vesilind) under certain conditions.



Representation of layered approach used in the Double Exponential settler model.

Bulk flux terms are given by the product of the up or downflow velocity and layer concentration, i.e.:

$$J_{BULK} = \frac{Q_I}{A} \cdot X_i \text{ (above feed layer)}$$

$$J_{BULK} = \frac{Q_R}{A} \cdot X_i \text{ (below feed layer)} \quad (7)$$

where Q_I = plant influent flow, Q_R = settling tank uderflow rate, A = settling tank cross-sectional area, and X_i = suspended solids concentration in layer *i*.

An important aspect of this model is that the gravity flux out of a layer is not allowed to be greater than the gravity flux that can be transmitted out of the next layer. This overcomes difficulties with "solids trapping" that can occur under certain loading conditions in unrestricted flux models like the Modified Vesilind model. Once again, gravity flux terms are given by the product of settling velocity and layer concentration but incorporating the flux limit, i.e.:

$$J_{S,i} = \min\left(V_{S,i} \cdot X_i, V_{S,i+1} \cdot X_{i+1}\right) \quad (8)$$

where $V_{S,i}$ = settling velocity in layer *i* and X_i = suspended solids concentration in layer *i*.

Sludge Settling Velocity

The Double Expontential settling tank model uses the settling velocity function proposed by Takacs *et al.* (1991). The settling velocity function is applicable in regions of flocculent and hindered settling, and is given by:

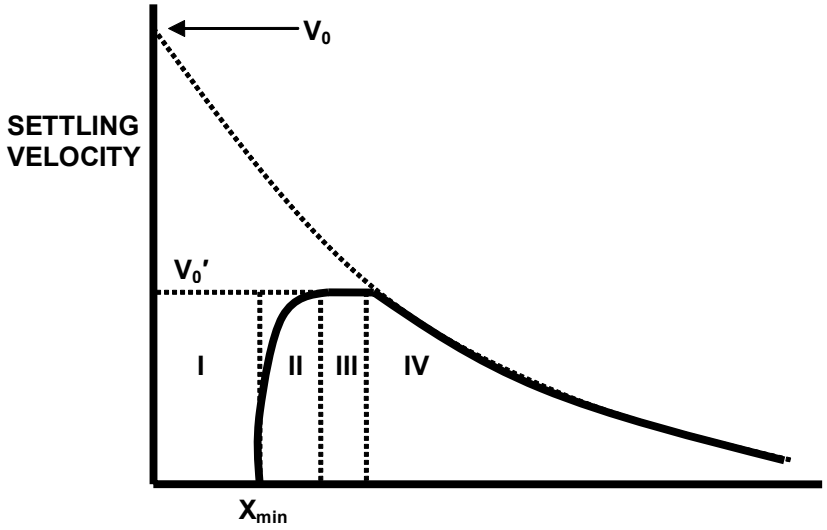
$$V_{S,i} = V_0\ e^{-K_h X_i^*} - V_0\ e^{-K_f X_i^*} \quad (9)$$

where V_0 = maximum Vesilind settling velocity, K_h = hindered zone settling parameter, K_f = flocculent zone settling parameter, and $X^*_i = X_i - X_{min}$ where X_i is the total suspended solids concentration in layer *i* and X_{min} is the minimum attainable suspended solids concentration.

The minimum attainable suspended solids concentration in a layer (X_{min}) is calculated as a fraction of the settling tank influent solids concentration:

$$X_{min} = f_{ns} \cdot X_{FEED} \quad (10)$$

Users may specify a maximum value for the X_{min} parameter (the BioWin default for this parameter is 20 mg/L). Another important user-specified parameter for the Double Exponential settling tank model is the maximum practical settling velocity, V_0'. This parameter sets the maximum gravitational settling velocity that is attainable by the solids. A graphical representation of the Double Exponential settling velocity function is shown in the figure below [note that this figure is meant to be conceptual – it is not necessarily to scale (e.g. region I is disproportionately large in the figure)].



The Double Exponential settling velocity function.

The Double Exponential settling velocity function consists of four regions:

1. In region I, the settling velocity is zero since the suspended solids concentration reaches the minimum attainable suspended solids concentration.
2. In region II, the settling velocity increases with suspended solids concentration since it is strongly influenced by the flocculent nature of the solids – the behavior of this zone is strongly influenced by the value selected for K_f.
3. In region III, settling velocity is independent of suspended solids concentration since it is hypothesized that solids particles have reached a maximum attainable size. The settling velocity in this region is set by the maximum practical settling velocity, V_0'.
4. In region IV, hindered settling becomes the dominant process and the settling velocity function reduces to the Vesilind function. The behavior of this zone is strongly influenced by the parameter K_h.

References and Further Reading

Bye, C. M. and Dold, P. L. (1999) Evaluation of correlations for zone settling velocity parameters based on sludge volume index-type measures and consequences in settling tank design, *Water Environment Research*, **71**, 1333-1344.

Bye, C.M. and Dold, P.L. (1998) Sludge volume index settleability measures: effect of solids characteristics and test parameters. *Water Environment Research*, **70**, (1), 87-93.

Takacs, I., Patry, G.G., and Nolasco, D. (1991) A dynamic model of the clarification-thickening process. *Wat. Res.* **25**(10), 1263-1271.

Vitasovic, Z. (1989) Continuous settler operation: a dynamic model. In: *Dynamic Modeling and Expert Systems in Wastewater Engineering*. (G G Patry and D Chapman: eds.), Lewis Publishers, Inc., Chelsea, Michigan.

Modeling of pH in BioWin

Importance of pH Modeling

It has been recognized from the early stages of wastewater process modeling that pH is an important factor in simulating the performance of biological wastewater treatment processes, including activated sludge and anaerobic digestion. The pH impacts the species distribution of the weak acid systems (carbonate, ammonia, phosphate, acetate, propionate, etc.) present in the process. This in turn dictates the rate of many of the biological and physico-chemical phenomena occurring in these systems. For example, (1) biological activity, that can be severely limited outside an optimal pH range, (2) chemical precipitation reactions when metal salts such as alum or ferric chloride are added for chemical P removal, (3) spontaneous precipitation of magnesium and calcium phosphates (struvite, HDP, HAP), and (4) stripping of ammonia at high pH. It is difficult to model pH because the underlying components and reactions are so fast and complex. The approach to date in activated sludge models has been to track alkalinity changes instead, and use that as a pseudo indicator of potential pH instability problems. This approach assumes that the pH remains approximately constant and is in a region where it does not impact biological activity. Another disadvantage of using alkalinity is that it offers no means for modeling physico-chemical phenomena such as precipitation. In systems with significant volatile fatty acid concentrations, such as acid fermenters or digesters or in systems where significant gas transfer may occur, the predicted alkalinity may not be a good indicator of steady pH conditions. Calculation of the pH must consider the concentrations of strong acids and bases, the dissociation states of the weak acid, carbonate and phosphate systems, chemical precipitation reactions, and potential stripping of components involved in the acid-base systems such as ammonia and carbon dioxide. All of these processes can be described using a kinetic approach (Musvoto *et al*., 2000), but the rates of many of the reactions involved typically are four to twenty orders of magnitude larger than typical biological rates. As a result, calculation of the pH using a kinetic-based model will significantly reduce simulation speed. BioWin uses a mixed kinetic/equilibrium based approach to minimize the negative impact on simulations speed. This approach is applicable across a wide range of biological treatment process models (i.e. ASM-series for activated sludge, ADM1 for anaerobic digesters, etc.).

Model Description

The pH model is based on the following elements:

1. Equilibrium modeling of the phosphate, carbonate, ammonium, volatile fatty acid systems and typical strong ions in wastewater (Mg^{2+}, Ca^{2+}, NO_3^-, etc.).
2. Incorporation of activity coefficients based on the ionic strength of the solution.
3. Gas-liquid transfer of ammonia and carbon dioxide.
4. Biological activity affecting compounds included in the model (e.g. CO_2 and many others)

Equilibrium expressions for the acid-base systems included in the model are shown in Table 1. These equlibria represent the predominant acid-base systems occurring in wastewater treatment systems.

All of the equilibrium expressions in Table 1 are expressed in terms of active concentrations rather than molar concentrations. The interaction of ions in solution causes a deviation from ideal behaviour whereby the activity of the ions in equilibrium reactions is less than expected from the molar concentrations. To account for this behaviour, the molar concentration of the ions is reduced by a factor known as the activity coefficient. The reduced ionic concentration is called the active concentration, as determined in the following expression:

$$(X_i) = f_i[X_i] \quad (1)$$

where

(X_i) = active concentration of Xi

$[X_i]$ = molar concentration of ion Xi

f_i = activity coefficient of ion Xi

Table 1: Acid-base equilibrium expressions included in the general pH model

System	Equilibrium Expression	Equilibrium Constant @20oC
Water	$\left(H^+\right)\left(OH^-\right) = K_w$	6.867×10^{-15}
Carbonic acid	$\frac{\left(H^+\right)\left(HCO_3^-\right)}{\left(H_2CO_3^*\right)} = K_{iCO_3\ 1}$	4.14×10^{-7}
Carbonic acid	$\frac{\left(H^+\right)\left(CO_3^{2-}\right)}{\left(HCO_3^-\right)} = K_{iCO_3\ 2}$	4.201×10^{-11}
Acetic acid	$\frac{\left(H^+\right)\left(CH_3COO^-\right)}{\left(CH_3COOH\right)} = K_{iAc}$	1.754×10^{-5}

Propionic acid	$\frac{\left(H^+\right)\left(CH_3CH_2COO^-\right)}{\left(CH_3CH_2COOH\right)} = K_{iPr}$	1.318×10^{-5}
Phosphoric acid	$\frac{\left(H^+\right)\left(H_2PO_4^-\right)}{\left(H_3PO_4\right)} = K_{iPO_4\ 1}$	7.452×10^{-3}
Phosphoric acid	$\frac{\left(H^+\right)\left(HPO_4^{2-}\right)}{\left(H_2PO_4^-\right)} = K_{iPO_4\ 2}$	6.103×10^{-8}
Phosphoric acid	$\frac{\left(H^+\right)\left(PO_4^{3-}\right)}{\left(HPO_4^{2-}\right)} = K_{iPO_4\ 3}$	9.484×10^{-13}
Ammonium	$\frac{\left(H^+\right)\left(NH_3\right)}{\left(NH_4^+\right)} = K_{iNH_3}$	3.966×10^{-10}

Activity coefficients are estimated in the BioWin pH model using the Davies equation, which is a simplification of the extended Debye-Hückel law. The activity coefficient (f_i) for each ion *i* in solution is determined as follows (Loewenthal and Marais, 1976):

$$\log f_i = -0.5 \cdot Z_i^2 \cdot \left(\frac{\sqrt{\mu}}{1+\sqrt{\mu}} - 0.2\mu \right) \quad (2)$$

where

Z_i = ionic charge of ion Xi

μ = ionic strength of solution

Note that because the deviation from ideal solution behaviour is caused by electrostatic attraction between ions, the activity coefficient of a neutral species in solution (i.e. $H_2CO_3^*$) is 1.

The expression for ionic strength is as follows:

$$\mu = 0.5 \sum_{i=1}^{n} [X_i] Z_i^2 \quad (3)$$

where

n = the number of ionic species in solution

Since the overall charge of the solution must be neutral, the sum of the concentrations of the positively charged ions in solution must equal the sum of the negatively charged ion concentrations. This charge balance relationship is expressed for the pH model as follows:

$$\begin{aligned}&[H^+] + [NH_4^+] + 2[Mg^{2+}] + 2[Ca^{2+}] + V_{Me}[Me] + 2[MeH_2PO_4^{2+}] + [MeHPO_4^+] + [Cations^+] = \\ &[OH^-] + [H_2PO_4^-] + 2[HPO_4^{2-}] + 3[PO_4^{3-}] + [HCO_3^-] + 2[CO_3^{2-}] + \\ &[CH_3COO^-] + [CH_3CH_2COO^-] + [NO_3^-] + [Anions^-]\end{aligned} \quad (4)$$

In addition to the ionic species shown in Table 1, there are a number of other ions likely to occur in significant concentrations in wastewater treatment systems, and these have been included in the charge balance. Calcium (Ca^{2+}) and Magnesium (Mg^{2+}) will be present in the natural source waters and general variables have also been added for cations ($Cations^+$) and anions ($Anions^-$) which can be used to account for additional species not included separately (for example, modeling addition of strong acids or bases - Na^+, Cl^-, etc.). In addition to naturally occurring ions such as Mg and Ca the charge balance also includes a general term for metal species (Me) with charge V_{Me}, which have been included to allow for modeling of chemical phosphorus removal. (See the **Ferric or Alum Precipitation** section of the *Process Model Formulation* chapter).

BioWin state variables track the total species concentrations for all the appropriate species thus the set of equations described by equations (1) through (4), the dissociation expressions in Table 1 and individual component material balances may be solved simultaneously to determine the individual ion concentrations, pH and ionic strength. This allows material balances to be calculated for each of the total species concentrations. For example, a material balance for total dissolved inorganic carbon in a reaction vessel is written as follows:

$$\frac{dM_{CO2t,L}}{dt} = Q_{L,i}S_{CO2t,i} - Q_{L,o}S_{CO2t,o} - (CO_2\,\text{Stripping}) + (\text{Net Reaction}) \quad (5)$$

where

$\text{Subscript},i$	= influent
$\text{Subscript},o$	= effluent
$M_{CO2t,L}$	= mass of dissolved total inorganic carbon
S_{CO2t}	= total dissolved inorganic carbon concentration
Q_L	= liquid flowrate
Net Reaction	= biological reaction rate

$$CO_2\,\text{Stripping} = -k_{L,CO2} \cdot A_{GT} \cdot (S_{CO2,sat} - [H_2CO_3^*]) \cdot V_L \quad (6)$$

where

k_L	= liquid phase mass transfer coefficient

A_{GT} = specific interfacial area for gas transfer

S_{CO2sat} = saturation dissolved CO_2 concentration

$[H_2CO_3^*]$ = undissociated carbonic acid concentration

The **Net Production by Reaction** term accounts for biological generation (e.g. from oxidation of organics by heteretrophs) and consumption (e.g. by autotrophs). The saturation concentration (S_{CO2sat}) in the "Gas Stripping" term is calculated from a Henry's law relationship at the system temperature and pressure. A simplification of the model assumes that the gas phase concentration of the component is constant (i.e. atmospheric concentration for CO_2, zero for ammonia), and therefore the saturation concentration of the dissolved component is constant for a given temperature. However, for many systems (e.g., anaerobic digesters), a material balance is also required for the gas phase. For the carbon dioxide component, the gas phase material balance is written as follows:

$$\frac{dM_{CO2,G}}{dt} = Q_{G,i} G_{CO2,i} - Q_{G,o} G_{CO2,o} + (CO_2 \text{Stripping})$$

where

$M_{CO2,G}$ = mass of CO_2 in the gas phase

Q_G = gas flowrate

G_{CO2} = gas phase carbon dioxide concentration

Similar material balances (gas and liquid phase) are required for the total ammonia concentration. For the other ionic species in solution (i.e. volatile acids, phosphate, etc.), there is no stripping and thus material balances for these components are written only for the liquid phase.

Alkalinity determination

Figure 1 shows a logarithmic-concentration versus pH diagram for the carbonate system. This diagram was generated with the BioWin pH model by simulating a system with a total dissolved inorganic carbon concentration of 10 mmol/L, and successively adding an increasing amount of anion to the system to change the pH. The model determines alkalinity by noting that at the H_2CO_3* equivalence point $[H^+] = [HCO_3^-]$. This additional equation can then be used to solve the carbonate equilibrium explicitly to determine the $[HCO_3^-]$ concentration at the equivalence point (and consequently the pH). From this a charge balance (at the equivalence point) can be used to calculate the amount of strong acid that would be required to move the solution from its current state to the H_2CO_3* equivalence point. As a result, the impact of all of the ionic species included in the general pH model is considered in the calculation of alkalinity. Note that the alkalinity is not *explicitly* related to the stoichiometry of the biological processes in the system when estimated with the pH model. Instead, it is related to concentrations of ionic species at the current system state.

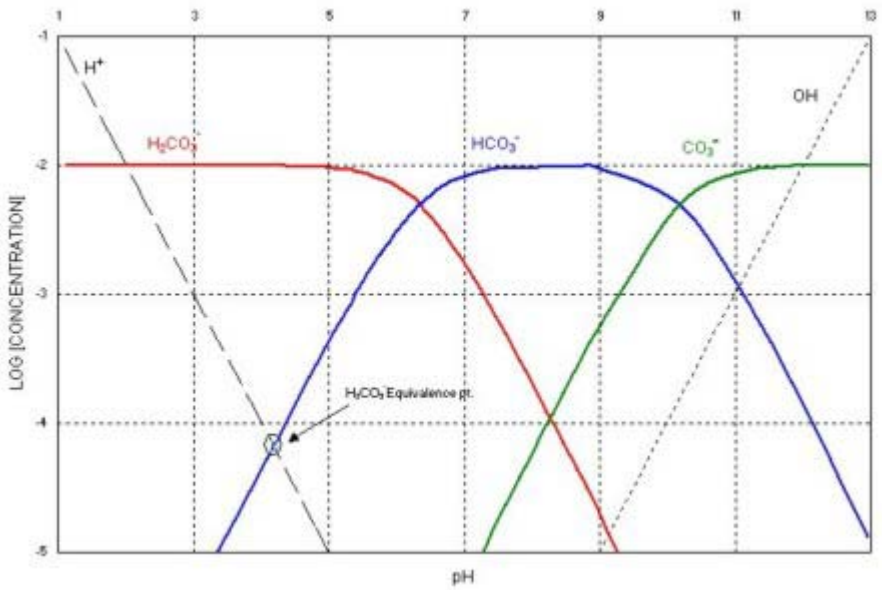


Figure 1: Logarithmic concentration versus pH diagram for the carbonate system as generated by the general pH model

Specifying pH and Alkalinity

In certain influent elements BioWin allows the user to specify the pH and alkalinity. Since both of these values are "calculated" from the current system state, it is necessary for BioWin to adjust the certain state variables to achieve the desired pH and alkalinity. In this case BioWin adjusts the total carbon dioxide, the anions and the cations until it achieves the desired pH and alkalinity. Note that it is possible to specify values that are not able to be achieved and BioWin will not warn you of the problem.

Biological inhibition due to pH

BioWin uses the calculated pH to determine the amount biological inhibition according to the following equation:

$$Inhibition = \frac{1 + 2\left(10^{\frac{\left(pH_{Low\lim it} - pH_{HighLimit}\right)}{2}}\right)}{1 + 10^{\left(pH - pH_{HighLimit}\right)} + 10^{\left(pH_{LowLimit} - pH\right)}} \quad (7)$$

where

pH = Calculated pH

$pH_{LowLimit}$ = Low pH limit for a particular organism

$pH_{HighLimit}$ = High pH limit for a particular organism

The user may select to turn of pH inhibition (essentially setting inhibition to 1.0) in all elements except the anaerobic digester element and the activated primary element. These units require a pH, but the user can choose to specify the pH that will be used in that element (i.e. either the calculated value, or a user specified value).

Note: Turning of pH inhibition (or specifying a pH in an element) does not stop BioWin from calculating the pH.

Examples

The following section contains a number of examples that highlight model performance and behavior.

Titration of Acids and Bases

Verification of the pH model included the simulation of various titrations of acids and bases in clean water. The simulation configuration (Figure 2) consisted of a variable volume reactor to represent a titration vessel, and an influent stream with a constant flow to represent the standard solution. The initial volume of the reactor was 50 L. The flowrate of the standard solution was 1 L/min. Table 2 summarizes simulated experimental conditions that represent the following types of titrations:

- Strong acid titrated with a strong base standard solution;
- Weak acid titrated with a strong base standard;
- Weak base titrated with a strong acid standard;
- Weak acid titrated with a weak base standard.

The table shows the concentrations of the reagents, and the components and concentrations used in the model to reflect those conditions. Note that in BioWin, the concentrations of reagents that are involved in the biological processes of the model (i.e. acetic acid and ammonia) are expressed in mg/L units. The concentrations of reagents that are primarily part of the acid base system (i.e. cations and anions), are expressed in units of meq/L.

Figures 3 to 6 show the results of the titration simulations compared to data from a standard chemistry text (Mortimer, 1975). These results verify that the acid-base equilibrium chemistry is correctly formulated in the pH model. The slight discrepancies between the actual and observed titration curves (for example, the equivalence point in Figure 4) are due to the incorporation of activity coefficients and the use of slightly different equilibrium constants in the general pH model.

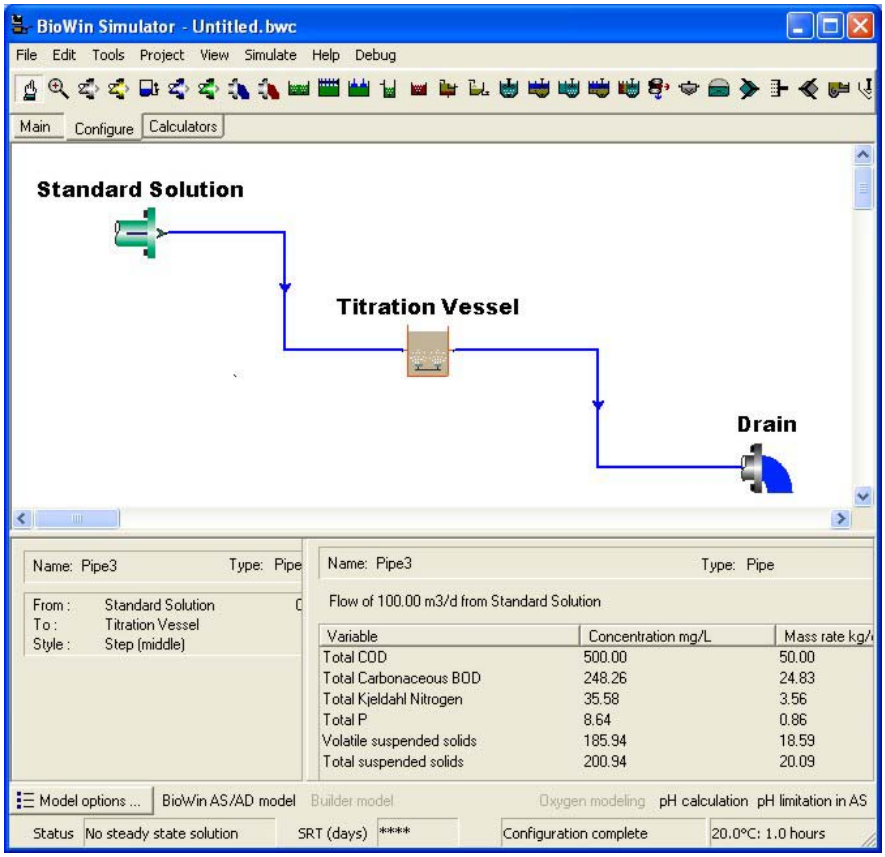


Figure 2: BioWin configuration for the simulation of various acid and base titration experiments

Table 2: Summary of simulated titration experiments

Titration Vessel Reagent	Standard Reagent	Titration Vessel Initial Concentrations	Standard Solution Concentrations
0.10 N HCl	0.10 N NaOH	[Anions-] = 100 meq/L	[Cations] = 100 meq/L
0.10 N CH_3COOH	0.10 N NaOH	[CH_3COOH] = 6000 mg/L	[Cations] = 100 meq/L
0.10 N NH_3	0.10 N HCl	[NH_3] = 1400 mg/L	[Anions] = 100 meq/L
0.10 N CH_3COOH	0.10 N NH_3	[CH_3COOH] = 6000 mg/L	[NH_3] = 1400 mg/L

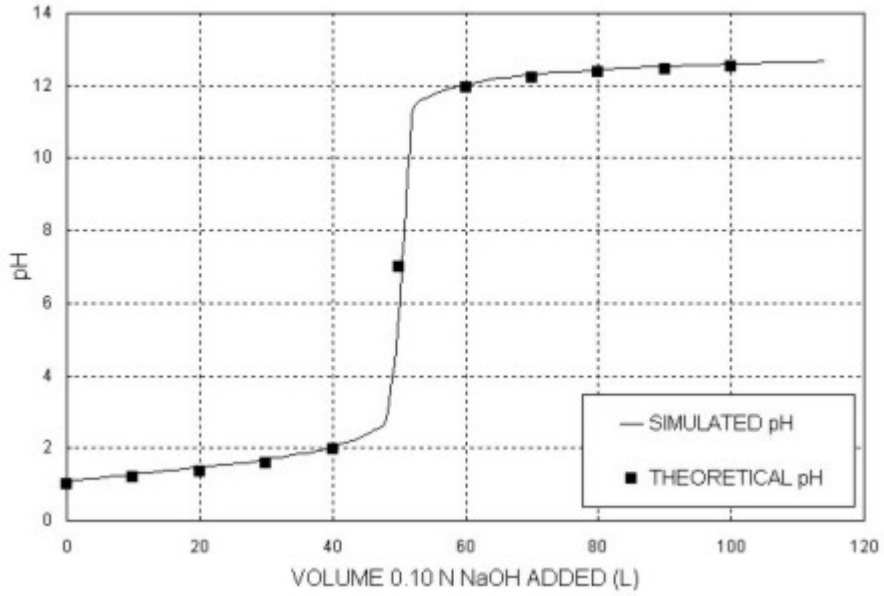


Figure 3: Results from simulation of 0.10 N HCL titrated with 0.10 N NaOH.

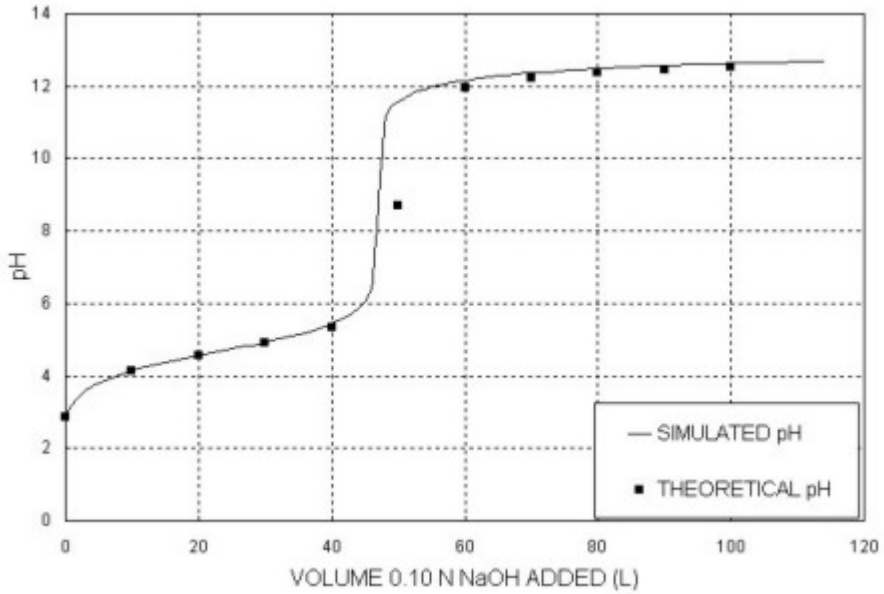


Figure 4: Results from simulation of 0.10 N CH_3COOH titrated with 0.10 N NaOH.

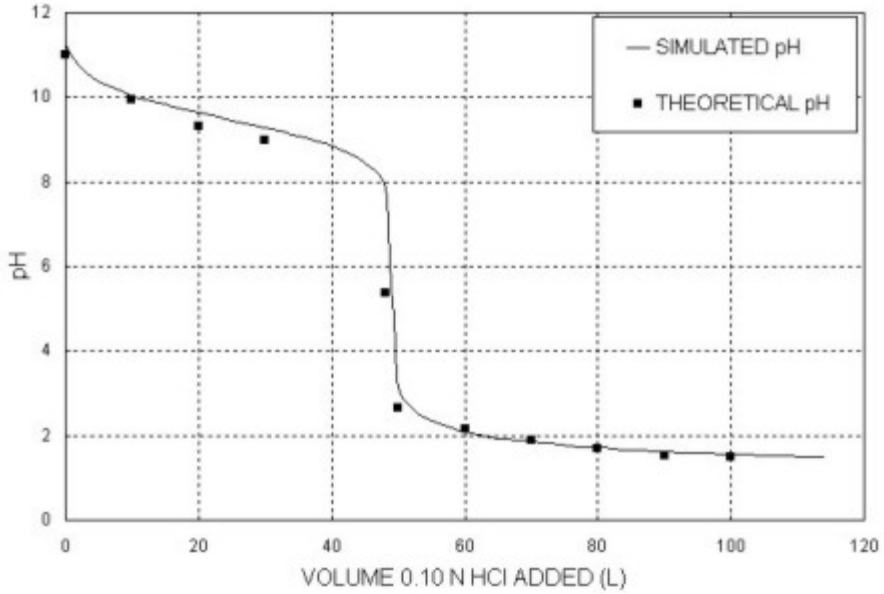


Figure 5: Results from simulation of 0.10 N NH_3 titrated with 0.10 N HCL.

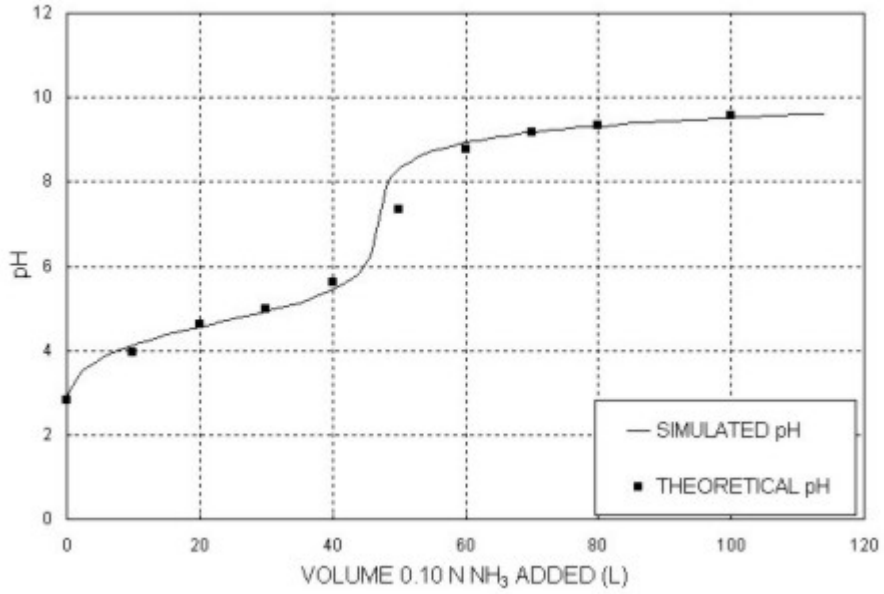


Figure 6: Results from simulation of 0.10 N CH_3COOH titrated with 0.10 N NH_3.

Considerations

A pH model is essential for reliable simulation of many important wastewater treatment operations, including:

1. Gas phase modeling which is important for modeling anaerobic digestion and precipitation processes. Calculation of gas transfer rates requires knowledge of the species ionization states and consequently the pH of the system.
2. Inhibition of biological activity at low and high pH.

3. Equilibrium-based, pH dependent modeling of aluminum and ferric dosing for phosphorus precipitation, including hydroxide sludge formation.
4. Kinetic-based, pH dependent modeling of the spontaneous precipitation of struvite and calcium phosphates. Accurate prediction of struvite precipitation also requires modeling of magnesium concentrations, both the soluble magnesium and that stored in organisms.

References

Batstone, D.J.; Keller, J.; Angelidaki, I.; Kalyuzhnyi, S.V.; Pavlostathis, S.G.; Rozzi, A.; Sanders, W.T.M.; Siegrist, H.; Vavlin, V.A. (2002). Anaerobic Digestion Model No.1, *IWA Scientific and Technical Report*.

Henze, M.; Grady, C.P.L.; Gujer, W.; Marais, G.v.R.; Matsuo, T. (1987a). Activated Sludge Model No. 1, *IAWPRC Scientific and Technical Report No. 1*, London, UK: International Water Association.

Henze, M.; Grady, C.P.L.; Gujer, W.; Marais, G.v.R.; Matsuo, T. (1987b) A general model for single sludge wastewater treatment systems. *Water Res.*, 21(5):505-515.

Jones, R. M.; Bye, C. M.; Dold, P. L. (2003) Nitrification Parameter Measurement for Plant Design: Experience with New Methods. Proceedings of the Water Environment Federation 76th Annual Technical Exhibition and Conference, Los Angeles, California, USA, October 11-15, *2003*.

Loewenthal, R.E.; Marais, G.v.R. (1976). *Carbonate Chemistry of Aquatic Systems: Theory and Application.* Ann Arbor Science, Ann Arbor Michigan.

Melcer, H.; Dold, P.L.; Jones, R.M.; Bye, C.M.; Takács, I.; Stensel, D.H.; Wilson, A.W.; Sun, P.; Bury, S. (2002). Methods for Wastewater Characterization in Activated Sludge Modeling. *Water Environment Research Foundation*, Alexandria, Virginia (in press).

Mortimer, C.E. (1975). *Chemistry a Conceptual Approach, 3rd Edition*, D. Van Nostrand, pp. 572-578.

Musvoto, E.V.; Ekama, G.A.; Wentzel, M.C.; Loewenthal, R.E. (2000). Extension and application of the three-phase weak acid/base kinetic model to the aeration treatment of anaerobic digester liquors. *Water SA*, **26**(4), pp. 417-438.

Musvoto, E.V.; Wentzel, M.C.; Loewenthal, R.E.; Ekama, G.A. (1997). Kinetic-based model for mixed weak acid/base systems. *Water SA*, **23**(4), pp. 311-322.

Perry, R.; Green, D.; (1985) Perry's Chemical Engineer's Handbook, Sixth Edition. McGraw-Hill Inc.

Gas-Liquid Mass Transfer

Introduction to Gas-Liquid Mass Transfer

Supply of oxygen constitutes a major operating cost for biological wastewater treatment systems. Emphasis on energy conservation has highlighted the need to develop effective methods for design and operation of aeration systems. Oxygen demand in activated sludge reactors varies with time, necessitating a varying oxygen supply rate to maintain the desired dissolved oxygen (DO) concentration.

In diffused air systems bubbles are distributed from diffusers at the base of the reactor. Mass transfer occurs between the rising bubbles and the mixed liquor. The transfer of oxygen from the gas to the liquid is required to supply the oxygen requirements for the biological process. A number of equipment and operational parameters interact to influence the efficiency and rate of transfer of oxygen; inter alia, diffuser pore size and density, and air flow rate. These parameters determine factors such as bubble size, the rate of bubble rise, the bubble residence time in the reactor, the fractional gas hold-up, the interfacial surface area available for mass transfer, the change in oxygen partial pressure in the rising bubbles and the degree of turbulence. Conditions in the mixed liquor also impact on the transfer; for example, temperature, ionic strength, presence of surface-active compounds, and solids concentration. Quantifying the impact of all these factors on the overall mass transfer behavior is very difficult.

This section explains briefly the gas-liquid mass transfer model used in BioWin. The discussion is structured in five parts:

Mass Transfer Theory: A number of theoretical approaches for quantifying mass transfer have been proposed. Researchers have used these theories as the basis for suggesting correlations for the design of aeration systems. These theories are reviewed to highlight the assumptions required by each approach.

BioWin Model for Mass Transfer:This section will provide details on the modeling approach used in BioWin.

Modeling Fine Pore Diffuser Performance:This section describes parameters available in BioWin to match fine pore diffuser performance.

Modeling Coarse Bubble Diffuser Performance:This section describes the parameters available in BioWin to match coarse bubble aeration performance.

Basic Parameters and Relationships:This section defines a number of commonly used terms and their formulation. The emphasis in this section is on oxygen mass transfer.

Conclusion:This section provides an example illustrating the differences between the methods used in BioWin and more traditional approaches.

Mass Transfer Theory

A number of theories have been proposed to explain the phenomenon of mass transfer in gas-liquid systems. These theories divide into two groups, describing the behavior under either laminar or turbulent flow conditions. Under laminar flow conditions, the mass transfer coefficient may be calculated directly since molecular diffusion prevails and the mathematics describing laminar flow and molecular diffusion are well defined. However, most practical applications of mass transfer involve turbulent flow where the flow regime is not well defined, and turbulent and molecular diffusion interact to determine mass transfer behavior. In bubble aeration systems such as those encountered in wastewater treatment applications turbulent flow conditions generally prevail.

Mass Transfer Theories Under Turbulent Flow Conditions

A number of theories have been proposed to describe mass transfer behavior under turbulent flow conditions. The principles of the more commonly used theories are discussed briefly below. Although there is some degree of overlap the theories generally fall into two major categories: those based on a rigid interface and those based on some form of surface renewal at the gas-liquid interface.

Rigid Interface Theories

Theories based on a rigid interface were the first to be proposed for modeling mass transfer behavior. These are conceptually similar to theories describing conductive heat transfer, and are based on the film theory of Lewis & Whitman (1924). In this approach it is proposed that the transfer process may be represented by molecular diffusion across a thin, static liquid film at the interfacial surface. The driving force for mass transfer is the concentration gradient between the interface and the outer edge of the thin liquid film (Figure 1). Using this representation, the liquid phase mass transfer coefficient, k_L, may be found from molecular diffusion theory (knowing the diffusivity) provided the effective film thickness can be determined. The effective film thickness is commonly considered to be a function of the flow conditions only. These models consequently predict that k_L depends directly on the molecular diffusivity of the solute $\mathcal{D}$. However experimental data has refuted this prediction. For example, Treybal (1981) reported that k_L is proportional to $\mathcal{D}^b$ where the exponent b varies from close to zero to 0.9.

The primary reason for the inability of rigid interface models to provide acceptable estimates of mass transfer coefficients is that there is movement of the liquid at the interface. Therefore the steady state situation depicted in (Figure 1) is not realistic and an alternate means for estimating the mass transfer coefficients is necessary. Nevertheless, mass transfer coefficients are still commonly called film transfer coefficients due to their historical origins in film theory.

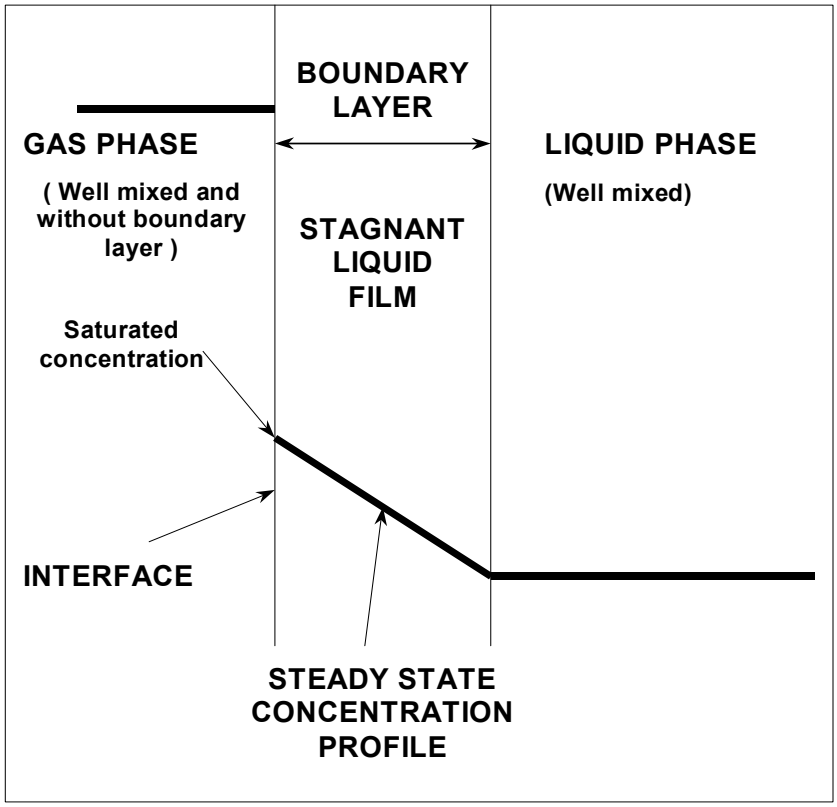


Figure1 : Schematic representation of film theory for mass transfer

Surface Renewal/Penetration Theories

The inability of rigid film theory to accurately predict mass transfer coefficients resulted in the proposal of a number of theories which incorporate some degree of unsteady state transfer. The most common of these theories are those which are based on the concept of surface renewal, where it is assumed that there is a continual interchange of liquid elements in contact with the gas phase. Theories based on the surface renewal concept, or theories combining rigid interface theory with surface renewal theory, have been widely accepted (Prasher & Wills, 1973). Perhaps the first surface renewal/penetration theory was proposed by Higbie (1935). Higbie's penetration theory has been used by a number of authors; for example, Calderbank (1959); Calderbank & Moo-Young (1961); Meijboom & Vogtlander (1973); Akita & Yoshida (1974); Kiode et al. (1976); and Popel & Wagner (1991). Modified surface renewal/penetration models have subsequently been presented and used by a number of authors (including Danckwerts 1951; Dobbins 1956; Marchello & Toor, 1963; Harriot, 1962; Sano et al. 1974; Hughmark, 1967; Treybal, 1981). Most currently accepted correlations for mass transfer parameters make use of one or more of these theories.

The mass transfer theories presented above are useful in determining the relationship between the mass transfer coefficient and other parameters involved in transfer. However they do not, in general, provide a direct means of predicting the transfer coefficient because they require the evaluation, or estimation of one or more of the hydrodynamic parameters in the model (exposure time, surface renewal rate, film thickness, or eddy depth). Evaluation of the hydrodynamic parameters is most often achieved through analysis of experimental mass transfer data (Prasher & Wills, 1973).

BioWin Model for Mass Transfer

The mass transfer theories proposed suggest several possible mechanisms controlling the phenomenon of mass transfer in bubble aerated aqueous systems. Determination of which mechanism most closely represents the actual transfer process must be based on experimental observations. Since direct measurement of the liquid phase mass transfer coefficient is not usually possible, an average liquid phase mass transfer coefficient is generally inferred from experimental determinations of the overall mass transfer coefficient.

The overall mass transfer coefficient, k_La_L, may be determined from a mass balance on the experimental data. Simultaneous mass balances for a component should be conducted on both the gas and liquid phases. This is necessary to account for the depletion of oxygen in the bubbles and mass transfer of other solutes between the liquid and gas phases (e.g. carbon dioxide). In general, to simplify the calculation procedure, certain assumptions are made regarding factors such as those mentioned above, as well as other factors such as the mixing conditions in both phases. Material balances can be written for each of the species concentrations (with gas phase interactions) by assuming that:

1. The gas phase in the reactor is completely mixed – that is the concentration of the components in the bubbles is the same for all bubbles and uniform within each bubble. [A similar, but more complicated model could be developed assuming plug flow of the gas through the liquid in the control volume.]
2. The liquid phase is completely mixed – that is the liquid phase concentrations are uniform through out the control volume.
3. The gas hold up is constant.

These material balances can be expressed as follows:

For the liquid phase:

$$\frac{dM_{j_T}}{dt} = \left[Q^{in} \cdot C_{j_T}^{in} - Q \cdot C_{j_T}\right] + \left[\alpha F \cdot \left(k_L a_L\right)_j \cdot (C_{\infty,j}^{*} - C_{j_{UN}}) \cdot V_L\right] + Rx_j \qquad (1)$$

For the gas phase:

$$\frac{dm_j}{dt} = \left[q^{in} \cdot c_j^{in} - q \cdot c_j\right] - \left[\alpha F \cdot \left(k_L a_L\right)_j \cdot (C_{\infty,j}^{*} - C_{j_{UN}}) \cdot V_L\right] \qquad (2)$$

where

M_{j_T} = Mass of component j (total of all ionization states) in the liquid phase

Q^{in} = Liquid flow into the control volume

Q = Liquid flow out of the control volume

$C_{j_T}^{in}$ = Influent liquid concentration of component j (total of all ionization states)

C_{j_T} = Effluent liquid concentration of component j (total of all ionization states)

$\left(k_L a_L\right)_j$ = Overall liquid phase mass transfer coefficient for component j

$\alpha = \frac{(k_L a_L)_{\text{Process Water}}}{(k_L a_L)_{\text{Clean water}}}$ = Ratio of process water to clean water both for a new, clean diffuser

$F = \frac{(k_L a_L)_{\text{Service time}}}{(k_L a_L)_{\text{New diffuser}}}$ = Ratio of process water after a given time in service to process water for a new clean diffuser

$C^*_{\infty,j}$ = Saturated concentration of component j at the gas/liquid interface

$C_{j_{UN}}$ = Bulk liquid concentration of unionized component j

V_L = Volume of the liquid phase

Rx_j = Net production of component j by reaction

m_j = Mass of component j in the gas phase

q^{in} = Gas flow into the control volume

q = Gas flow out of the control volume

c_j^{in} = Influent gas concentration of component j

c_j = Effluent gas concentration of component j

The "Net Production by Reaction" term accounts for biological generation and consumption of the component of interest.

Note : A simplification of the model assumes that the gas phase concentration of the component is constant, and therefore the saturation concentration of the dissolved component is constant for a given temperature. In this case a gas phase material balance is not required however, for many systems (e.g., anaerobic digesters), a material balance is also required for the gas phase. In BioWin the user has the option to decide whether or not to include a gas phase material balance for bioreactor (for digesters a gas phase material balance is always required). If the user elects to omit the gas phase material balance then BioWin will use the "off-gas" concentrations specified by the user on the "Aeration" parameters tab. For Surface aerators, BioWin uses the "Supply gas" concentrations to determine the saturation concentration.

Application of the model described above requires a method for determining the saturation concentration, and the overall mass transfer coefficient, for each species subject to gas transfer.

Determination of the Steady State Saturation Concentration at Field Conditions

The required saturation concentration at field conditions may be estimated using a Henry's law expression of the form:

$$C_j^* = \frac{p_j}{H_j} \tag{3}$$

where

p_j = Partial pressure of component j

H_j = Henry's law constant

The Henry's law constant is a function of the temperature. In BioWin the partial pressure is calculated, at 1 atmosphere, from the dry mole fraction of the solute in the gas phase (corrected for humidity) and the temperature.

Note: If the gas phase is not modelled then BioWin uses off-gas concentrations specified to calculate the saturated concentration, except in surface aerated vessel in which BioWin uses the supply gas concentrations.

The impact of diffuser submergence on pressure is calculated from:

$$\Omega' = \frac{P_{in\ situ}}{P_{Std}} = \frac{P_{Field} + \left(\rho_{water} \cdot d_e \cdot g\right) - P^v_{water}}{P_{Std} - P^v_{water}} \tag{4}$$

where

$P_{in\ situ}$ = Average pressure in the liquid phase

P_{Std} = Standard atmospheric pressure

P_{Field} = Atmospheric pressure at the liquid surface at field conditions

P^v_{water} = Saturated water vapour pressure at field liquid temperature

ρ_{water} = Density of water at field liquid temperature

d_e = Fractional equivalent saturation depth

g = Gravitational acceleration constant

The saturated vapour pressure of water at temperature T°C can be obtained from tables, or estimated (for T > 0) using the following correlation:

$$P^v_{water} = 0.73402303 \cdot \left(1.058297462\right)^T \tag{5}$$

Note: For reactors with surface aeration the diffuser submergence term, evaluates to zero and the correction reduces to a field pressure correction only.

The steady state dissolved component concentration may now be determined from:

$$C^*_{\infty,j} = \beta \cdot \Omega' \cdot \frac{p_J}{H_j} \tag{6}$$

where

$$\beta = \frac{\text{Process water } C^*_\infty}{\text{Clean water } C^*_\infty}$$

Determination of the Overall Mass Transfer Coefficient at Field Conditions

The most appropriate method for estimating the overall mass transfer coefficient depends on the aeration method.

Determination of the Overall Transfer Coefficient for Oxygen in Surface Aerated Vessels

In systems aerated using a surface aeration device BioWin allows the user to enter the "Device Standard oxygen transfer rate" (mass transfer rate per unit power input at standard conditions). By specifying this value the user is essentially specifying the overall mass transfer coefficient for the device in clean water. BioWin uses the entered value to determine the overall mass transfer coefficient for surface aerators.

$$SOTR' = \frac{\left(k_L a_L\right)_{Oxygen} \cdot C^*_{\infty,Std}}{\Psi}$$

or (7)

$$\left(k_L a_L\right)_{Oxygen} = \Psi \cdot \frac{SOTR'}{C^*_{\infty,Std}}$$

where

$C^*_{\infty,Std}$ = Steady state dissolved oxygen concentration at 20°C and 1 atmosphere and the supply gas oxygen concentration (dry basis)

Ψ = Power input

Determination of the Overall Transfer Coefficient for Oxygen in Diffused Aeration Systems

Research has shown that the oxygen mass transfer coefficient, correlates well with the superficial gas velocity in diffused air systems. In BioWin the oxygen value is estimated using a correlation of the form:

$$\left(k_L a_L\right)_{Oxygen} = C \cdot U_{SG}{}^{Y} \tag{8}$$

where

$U_{SG} = \dfrac{q^{in}}{\text{Cross sectional area of tank}}$ = Superficial gas velocity

C, Y = Correlation parameters

Selection of the correlation parameters *C* and *Y* depends on the type and are discussed in more detail in the *Modeling Fine Pore Diffuser Performance*section.

Note: Correlation parameters for the mass transfer coefficient, $k_L a_L$, in BioWin are included in the model parameter editor on the **Diffuser** tab. This tab can be accessed by clicking the **Model Parameters** button on the **Operation** tab of an aerated bioreactor element.

Determination of the Overall Transfer Coefficient for Other Components

There is little information about the overall mass transfer coefficient for other components (NH_3, CO_2, etc) in wastewater treatment processes; consequently BioWin uses the overall mass transfer coefficient determined for oxygen to calculate the overall mass transfer coefficient of the other components. By noting that:

$$\frac{(k_L a_L)_j}{(k_L a_L)_i} = \frac{k_{L,j} \cdot a_L}{k_{L,i} \cdot a_L} = \frac{k_{L,j}}{k_{L,i}}$$

or (9)

$$(k_L a_L)_j = \frac{(k_L a_L)_i}{k_{L,i}} \cdot k_{L,j}$$

where

$k_{L,i}$ = Liquid phase mass transfer coefficient for component j

$a_L = \frac{A_{\text{Gas interface}}}{\text{V}_\text{L}}$ = Specific interfacial gas transfer area

BioWin allows the user to enter liquid phase mass transfer coefficient for all components except oxygen (which is assumed to be 12.96 m d^{-1} for the purpose of this calculation).

Modeling Fine Pore Diffuser Performance

This section describes the use of the BioWin diffused air correlation to model the performance of different fine bubble diffusers. In the section *Determination of the Overall Transfer Coefficient for Oxygen in Diffused Aeration Systems* the following correlation was introduced:

$$(k_L a_L)_{Oxygen} = C \cdot U_{SG}{}^{Y}$$

An example of a fit using this correlation, to full-scale experimental data for 9 inch ceramic domes is shown in Figure 2.

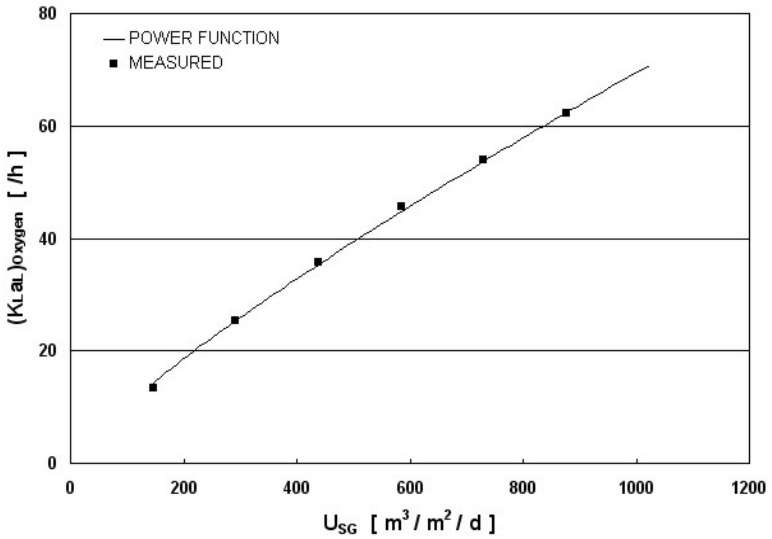


Figure 2 : Overall oxygen mass transfer coefficient as a function of superficial gas velocity at a diffuser density of 25%.

The figure demonstrates that the correlation can accurately predict the observed data. Manufacturers generally provide diffuser performance data in the form of a set of curves for a range of diffuser densities showing SOTE/depth (%/m or %/ft) versus air flow rate per diffuser. Typically, for fine bubble systems the curves start in the region of 8%/m (2.5%/ft) for low air flow rates, decrease with increasing air flow rate, and level off at higher air flows. An example is shown in Figure 3.

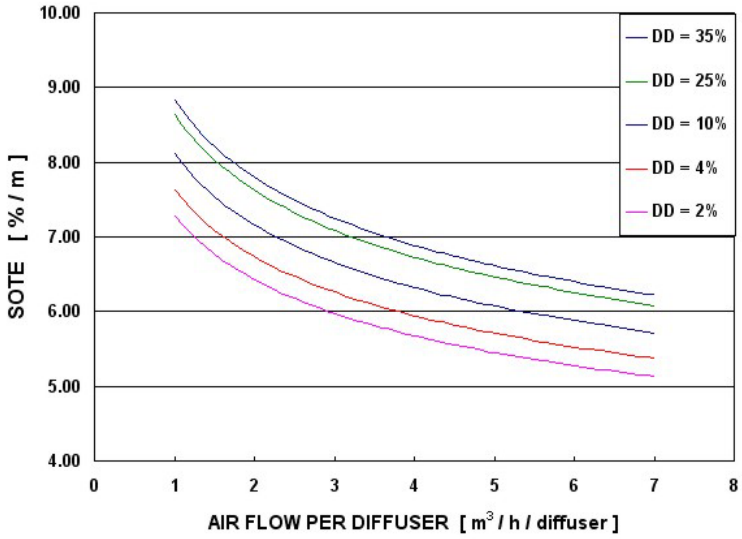


Figure 3 : Impact of air flow per diffuser on standard oxygen transfer efficiency

These data reflect the change in the mass transfer coefficient, with increasing air flow rate and diffuser density since SOTE is a function of the mass transfer coefficient.

Impact of Diffuser Density

The overall oxygen mass transfer coefficient for a given air flow per diffuser obviously increases with increasing diffuser density because there is a greater total

air flow. However, Figure 4 shows that the increase is not proportional to the increase in diffuser density.

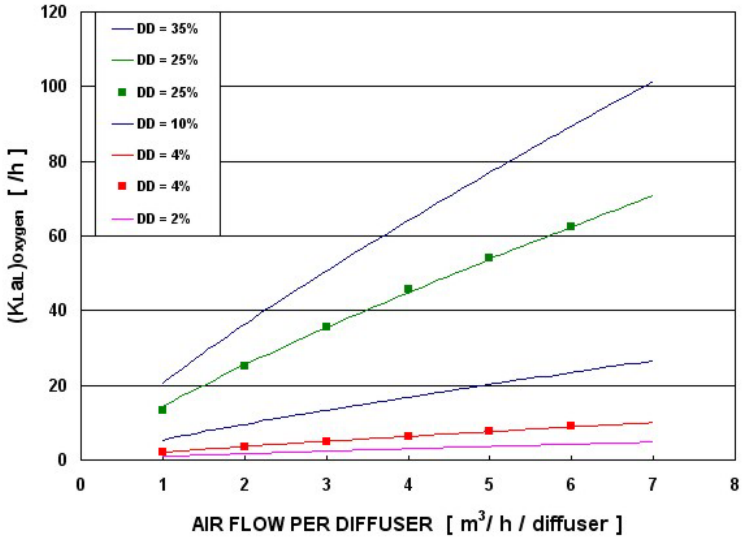


Figure 4 : Oxygen mass transfer coefficient versus air flow per diffuser for a range of diffuser densities.

In effect, the parameter *C* in Eq. 8 increases with diffuser density while *Y* is fixed for a specific diffuser type. Analysis of data for a number of different diffuser types and sizes showed that the parameter *C* in Eq. 8 can be related to the diffuser density as follows (see Figure 5): Relationship of correlation parameter *C* to diffuser density.)

$$C = K_1 \cdot DD\%^{0.25} + K_2 \qquad (10)$$

where

K_1 and K_2	= Correlation parameters with the following default values in BioWin $K_1 = 2.5656$ $K_2 = 0.0432$
DD%	= Percentage diffuser density see *Formulation of diffuser density, DD%*

Note: Correlation parameters for the mass transfer coefficient, k_La_L, in BioWin are included in the model parameter editor on the **Diffuser** tab. This tab can be accessed by clicking the **Model Parameters** button on the **Operation** tab of an aerated bioreactor element.

The exponent parameter value in equation was determined empirically. However, there may be some underlying theoretical basis for the value of 0.25. A theoretical analysis of aerated systems with separate mixing by Kawase and Moo-Young (1990) indicated that the *C* parameter value should be related to the power dissipation rate raised to the 0.25 power. In fine bubble diffuser systems power dissipation rate should be related to diffuser density for a given air flow per diffuser.

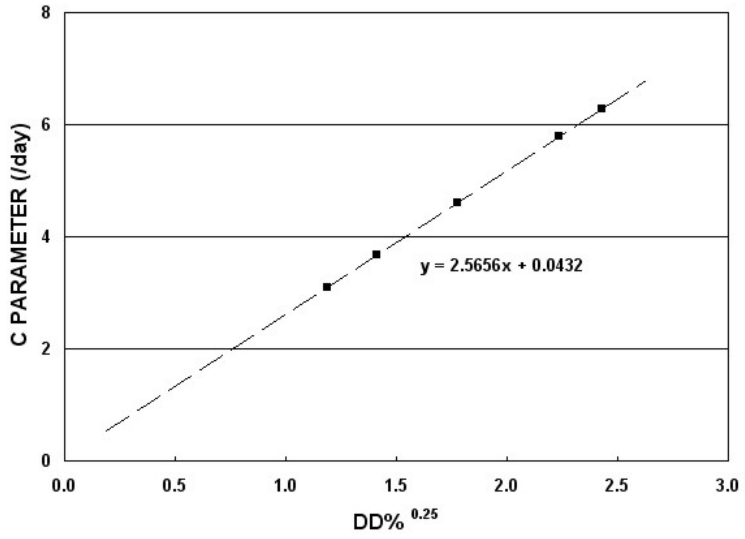


Figure 5 : Relationship of correlation parameter C to diffuser density.

Figure 6 illustrates the result of using Eq. 10 to determine *C* in Eq. 8. The figure demonstrates that a single parameter set can be used to predict SOTE (%/m) with changing air flow per diffuser and diffuser density (4 and 25%].

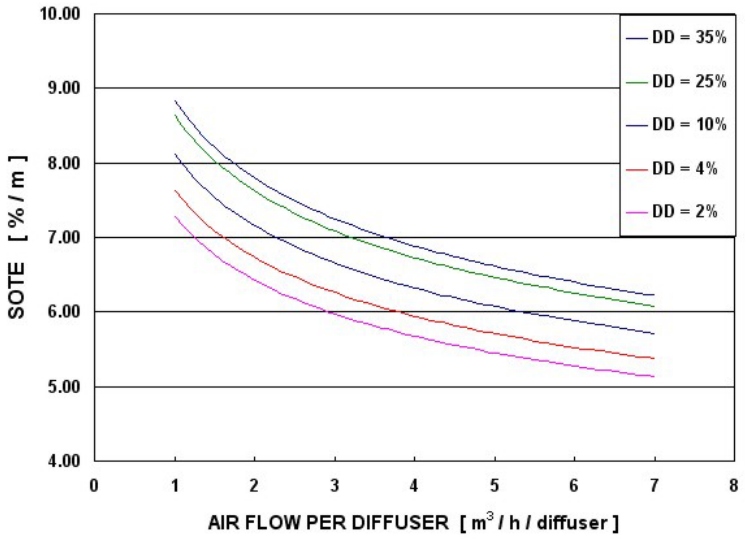


Figure 6 : Predictions of SOTE (% per unit depth) versus air flow rate per diffuser for two diffuser densities [ATAD = 100/DD%].

Note: An important consideration in the design of diffused air systems is that the overall mass transfer coefficient changes with changing air flow rate; that is, $k_L a_L$ is not "constant". Under dynamic loading conditions, over periods when the oxygen demand is high, and a higher air flow is required, there is a drop-off in transfer efficiency. This is an important factor when determining peak blower air delivery requirements.

BioWin allows the user to set up plots for bioreactors showing SOTE/depth (%/m or %/ft) versus air flow rate per diffuser [An example system including an SOTE plot can be downloaded from the EnviroSim Web site (www.envirosim.com)]. The chart is shown below. SOTE data are plotted for the diffuser density of the bioreactor in

question, and for four other user-selected diffuser densities. The example includes experimental data for a number of diffuser types]. The plot can be compared to manufacturer data, and used as a basis for selecting appropriate parameters (K_1 and K_2) for a particular diffuser type. Alternatively users can contact EnviroSim to obtain a spreadsheet to assist in parameter selection.

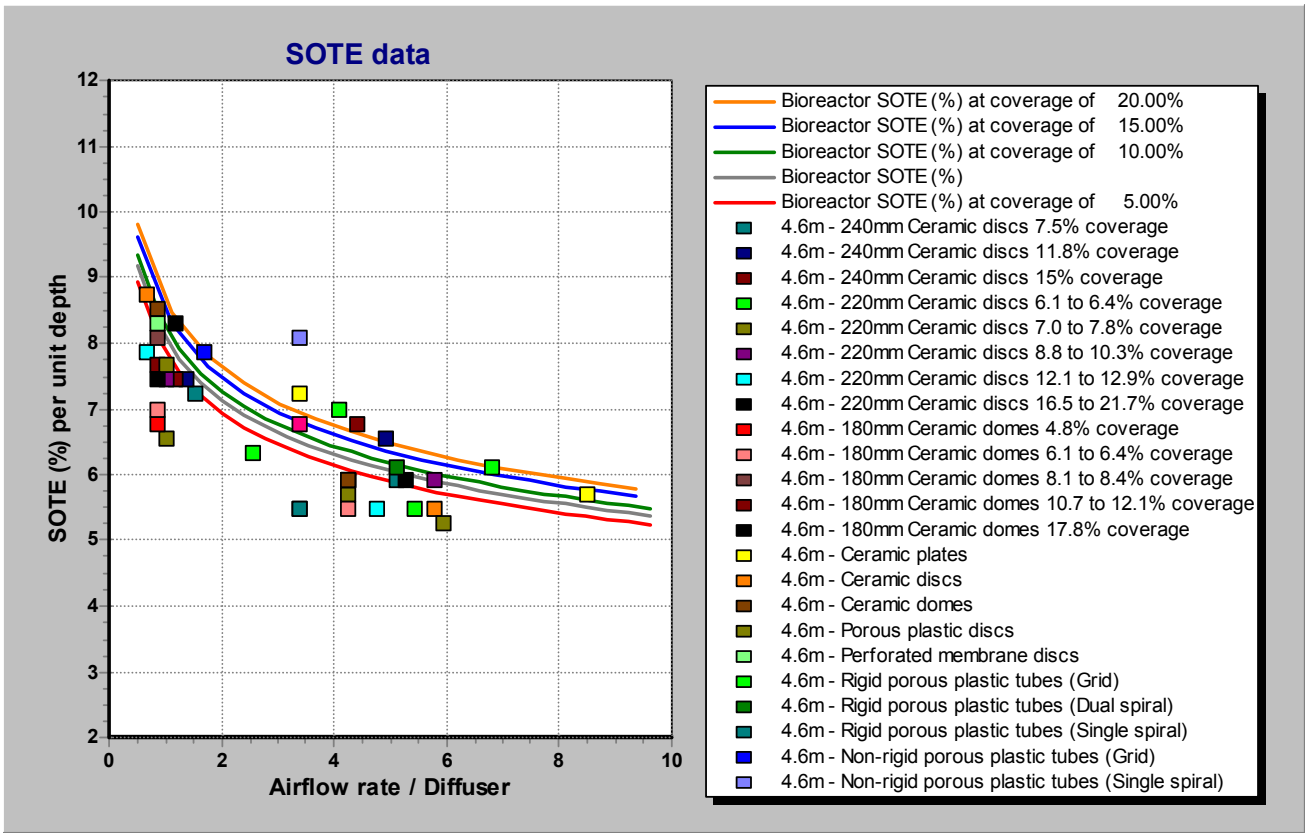


Figure 7 : BioWin SOTE plot to assist with correlation parameter calibration

Modeling Coarse Bubble Diffuser Performance

In general the performance of coarse bubble diffuser systems is very application specific and the concept of diffuser density and superficial gas velocity are not as useful as in fine bubble systems. Nevertheless, the BioWin correlation Eq. 8 can usually be used to adequately predict performance. For coarse bubble systems the impact of increasing superficial gas velocity is usually much less and consequently the correlation between superficial gas velocity and overall mass transfer coefficient is almost linear (the Y parameter approaches 1.0). The parameters K_1 and K_2 can be used to adjust the slope of the curve appropriately (generally K_1 is a small number reflecting the reduced impact of diffuser density).

Basic Parameters and Relationships

There are a number of parameters used to assist in describing and analysing the mass transfer behavior of a system that warrant definition or further discussion.

Formulation for SOTR

Standard Oxygen Transfer Rate (SOTR) is defined as the mass rate at which oxygen is transferred to clean water at 20°C and 1 atm surface pressure. In BioWin this is calculated as:

$$SOTR = \left(K_L a_L\right)_{Oxygen}^{20} \cdot \Omega_{1atm}'^{20} \cdot C_{\infty,Std}^{*} \cdot V_L \tag{11}$$

where

$C_{\infty,Std}^{*}$ = Steady state dissolved oxygen concentration at 20°C and 1 atmosphere and an oxygen mole fraction of 0.209 (dry basis).

Formulation of SOTE

The efficiency of aeration equipment usually is quantified in terms of the mass of oxygen transferred per mass of oxygen input in clean water tests (Standard Oxygen Transfer Efficiency, SOTE). Diffuser performance is often presented as a series of curves (for different diffuser densities and one diffuser submergence) showing SOTE (% per unit depth) versus air flow per diffuser.

$$SOTE = \frac{SOTR}{M_{O_2}} \cdot 100\% \tag{12}$$

where

M_{O_2} = Mass rate of oxygen supply

Formulation for OTR

The oxygen transfer rate (OTR) is the rate at which oxygen is actually transferred under field conditions.

$$OTR = \alpha F \cdot \left(k_L a_L\right)_j \cdot (C_{\infty,j}^{*} - C_{j_{UN}}) \cdot V_L \tag{13}$$

Formulation for OTE

The oxygen transfer efficiency (OTE) is the field efficiency of oxygen transfer.

$$OTE = \frac{OTR}{M_{O_2}} \cdot 100\% \tag{14}$$

Formulation of Diffuser Density, DD%

Diffuser density is defined in terms of the coverage:

$$DD\% = \frac{\#\ \text{diffusers} \cdot \text{area per diffuser}}{\text{bioreactor area}} \times 100 = \frac{1}{ATAD} \times 100 \quad (15)$$

where AT = Area of aeration tank.

therefore:

$$ATAD = \frac{AT}{AD} = \frac{100}{DD\%} \quad (16)$$

where AD = Total area of diffusers in aeration tank.

Bubble Size

Bubble size and gas/liquid interfacial area are important parameters in the analysis of mass transfer behavior. In a typical fine bubble diffused air system the size of individual bubbles is not uniform; rather the bubbles range in size.

Gas Hold-Up

The gas hold-up is an important parameter in mass transfer studies and is frequently used in correlations for bubble diameter or Sauter diameter and interfacial area. Gas hold-up is generally reported as the *fractional gas hold-up*; that is, the gas volume per unit dispersed phase volume. In BioWin when a gas hold up is specified it is specified as a percentage of the liquid volume. The *fractional gas hold-up* can be determined as follows:

$$\varepsilon = \frac{V_G}{V_D} = \frac{\phi/100}{\phi/100 + 1} \quad (17)$$

where

ε = Fractional gas hold-up

V_G = Volume of gas phase

V_D = Volume of dispersed phase

ϕ = BioWin gas hold-up percentage

Liquid Phase Mass Transfer Coefficient

The liquid phase resistance to mass transfer is the rate controlling process in most gas-liquid contacting operations (Treybal, 1981). The liquid phase mass transfer coefficient k_L is the parameter used to describe this resistance and consequently is a fundamental parameter in oxygen transfer. Unfortunately direct measurement of the transfer coefficient in bubble aeration systems is not possible and it is generally inferred from measurements of the overall mass transfer coefficient. This method

results in an average value for k_L, since it is conceivable that k_L varies even on the surface of a single bubble and probably from bubble to bubble. It should also be noted that calculation of k_L from the overall transfer coefficient assumes that the specific interfacial area can be represented adequately by a single value for the whole reactor (unless the overall transfer coefficient is found as a function of position in the reactor which is uncommon in experimental work).

Units for Air Flow

BioWin presents air flowrates in:

- m^3/hr
- ft^3/min (SCFM)

In each case the basis is 20°C, 1atm (14.7 psi, 101.325 kPa)

The units for the off-gas are m^3/day (in SI units) and ft^3/min (CFM) at field conditions.

Conclusion

The current approach for the design of aeration systems typically involves using a spreadsheet program to estimate peak, average and minimum air demand in each aerated zone of a system, based on the estimated oxygen demand. The peaks and minimums in different zones usually will not be coincident, so it is difficult to accurately estimate total air requirements, as well as time-varying requirements. The modeling approach in BioWin overcomes these problems through quantifying (1) the change in oxygen mass transfer coefficient (k_La_L) with changing air flow in each zone, and (2) accounting for different diffuser densities. BioWin quantifies time-and-space variations in oxygen demand, providing accurate estimates of time-varying air supply in different zones and hence more realistic estimates of total air supply needs. It also allows a comprehensive analysis of the interaction between diffuser density and oxygen transfer efficiency for the optimal design of aeration systems.

References and Additional Reading

EPA Design Manual for Fine Pore Aeration Systems (EPA/625/1-89/023)]:

Fairlamb, P.M. (1991). Bubble size and oxygen transfer in diffused air activated sludge systems. M.Eng. thesis, McMaster University, Department of Civil Engineering, Hamilton, Ontario.

Dold, P.L. and Fairlamb, P.M. (1990). Bubble size, size distribution and oxygen transfer in diffused air systems. *Proc. CSCE Conference*, Hamilton, Ontario (May), pp.576-594.

Johnson, T.L. and McKinney, R.E. (1994). Modeling full-scale diffused aeration systems. *Proc. ASCE Environmental Engineering Conference.*

Kawase, Y., and Moo-Young, M. (1990). Mathematical models for design of bioreactors: applications of Kolmogoroff's theory of isotropic turbulence. *The Chemical Engineering Journal,* 43, pp. B19-B41.

Randall, E.W., Goodall, C.M., Fairlamb, P.M., Dold, P.L. and O'Connor C.T. (1989). Method for measuring the sizes of bubbles in two- and three-phase systems. *J. Phys. E: Sci. Instrum*., 22, pp. 827-833.

Anaerobic Digestion Modeling

Introduction to Anaerobic Digestion Modeling

This section provides a general description of the modeling of anaerobic processes in BioWin particularly as they would occur in anaerobic digestion processes. The following subsections are included in this discussion.

Background on Model Development: This sub-section provides a brief literature review on anaerobic digestion modeling.

Model Formulation: This sub-section provides an overview of the structure and interactions of the model processes that predominate in an anaerobic digestion simulation.

Anaerobic Digestion vs. Activated Sludge: In BioWin, one comprehensive model for the biochemical and physical-chemical systems is used for the entire treatment plant. However, the behavior of a specific unit process element and the dominant reactions in that element are dependant on the process environmental conditions such as SRT, temperature, and pH. This sub-section describes how certain processes behave differently in anaerobic digestion vs. activated sludge.

Anaerobic Digester Performance Relationships: A series of simulations was conducted to examine the relationship of anaerobic digester performance to a number of process variables. This sub-section summarizes the results from the simulations.

Simulation of Sidestream Impact on BNR Processes: To investigate the impact of the digester on biological phosphorus removal behavior, published data from a municipal BNR demonstration study were examined. This sub-section discusses this investigation.

Background on Anaerobic Model Development

Early mechanistic mathematical models of anaerobic digestion considered only methanogenic reactions based on the assumption that methane production is the rate-limiting step in the process (Andrews, 1969; Andrews and Graef, 1971). An initial extension of this conceptual framework (Hill and Barth, 1977) included the reactions of "acid-formers" which considered both particulate hydrolysis and production of volatile fatty acids (VFAs). These early models represented the VFA concentration as one "bulk" component that was considered to be the sole substrate for the methanogens.

The role of hydrogen in the regulation of product distribution and consumption was a key development that formed the basis for many subsequent models of anaerobic

digestion (Mosey, 1983). This advancement allowed models to predict the formation of various fermentation products in addition to acetic acid, such as the higher acids propionic and butyric. In addition, methane production from both acetic acid and hydrogen could be included. A number of models were developed based on the "four population" framework of Mosey, including Costello et al. (1991) and Jones et al. (1992).

Most new models for anaerobic digestion include hydrogen regulation functions and are being presented using the same structured matrix approach as used for activated sludge models such as ASM1 and ASM2. Examples include the work of Massé and Droste (2000), and Bagley and Brodkorb (1999).

Important considerations in an anaerobic sludge digestion model when included in a comprehensive wastewater treatment plant simulator, include:

- Fate of influent biomass in the digester;
- Net release of ammonia and phosphate in the digester;
- Role of the anaerobic reactions in the activated sludge process.

The International Water Association recently has published the Anaerobic Digestion Model No. 1 (ADM1). Although this is a detailed and comprehensive model for anaerobic digestion processes, it has a number of limitations for application to plant-wide wastewater treatment plant simulation:

- ADM1 contains a set of state variables that is not consistent with standard activated sludge models (i.e. the ASM-series, the BioWin model and others). An "Interface Model" is required between the liquid and solids line if ADM1 is used to model the digester. Some of the activated sludge state variables must be "combined" into ADM1 state variables, and important information is lost on the fate of specific states.
- ADM1 does not include phosphorus as a state variable. The release of phosphorus in digesters is particularly important in simulating performance of BNR treatment plants.
- ADM1 does not include nitrogen release on organism decay and does not maintain a nitrogen mass balance. Ammonia release in the digester and return of the digester supernatant to the liquid line is an important consideration in a plant-wide simulation.

Model Formulation

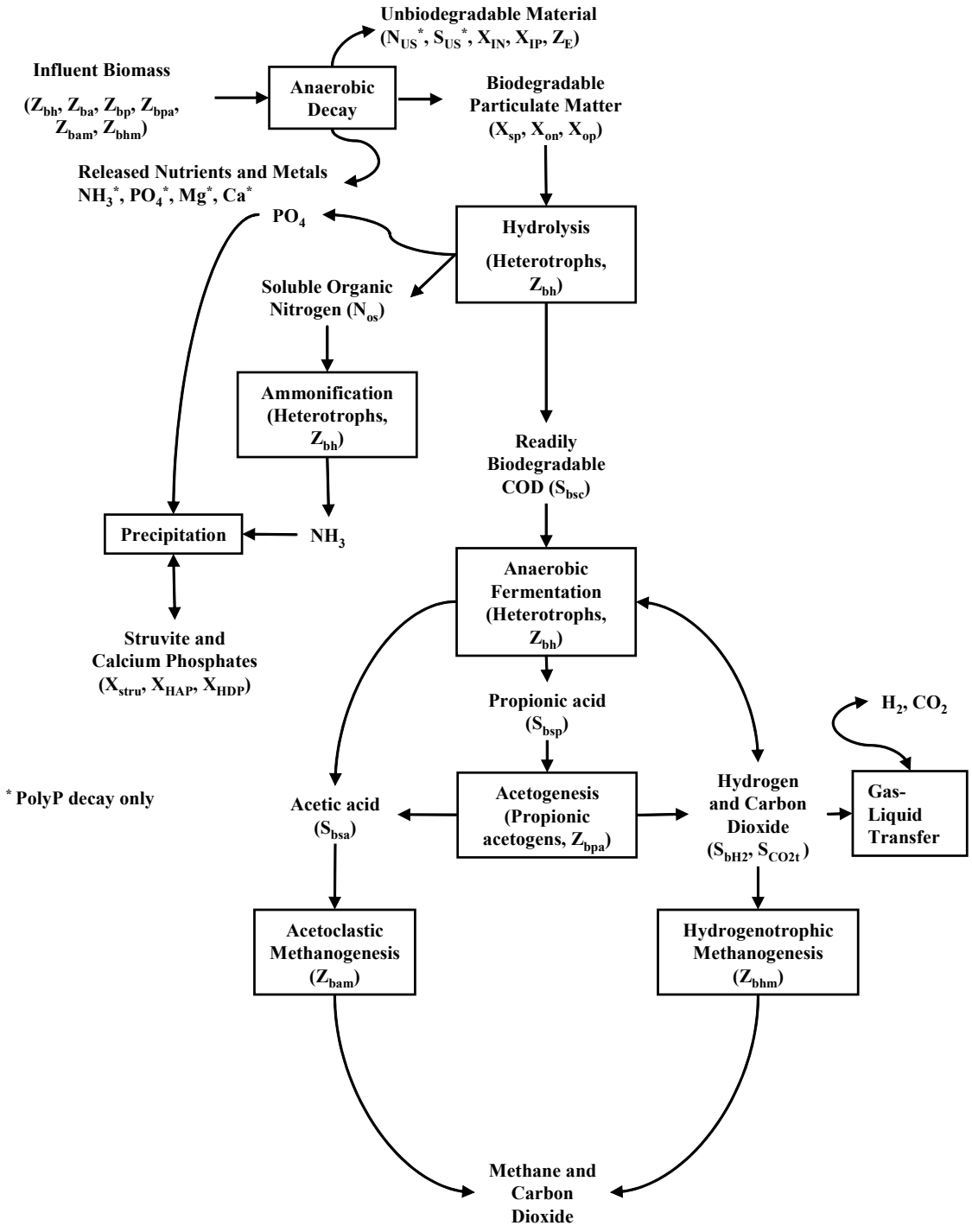
A key objective in formulation of the anaerobic processes in the model was to maintain a consistent set of state variables, stoichiometry, and process rate equations throughout the BioWin model. This allows BioWin to use one comprehensive model for the biochemical systems in the treatment plant – the behavior of a specific unit process element and the dominant reactions in that element being dependant on the process environmental conditions such as SRT, temperature, and pH.

The anaerobic degradation processes in the model are based on the "four population" model concept. A conceptual schematic of the model is shown in the figure below. The following points describe the key stages:

- Influent biomass (which would be present in significant quantities in waste activated sludge) undergoes anaerobic decay in the digester. The

process rates and stoichiometry are the same as for anaerobic decay in the activated sludge process. The products of decay include unbiodegradable organic (Z_E, S_{US}), nitrogen (X_{IN}, N_{US}), and phosphorus (X_{IP}) components. If polyP organisms (Z_{bp}) are present, the decay process also releases ammonia (NH_3), phosphate (PO_4-P), Magnesium (Mg) and Calcium (Ca).

- Hydrolysis of particulate matter is mediated by the (non-polyP) heterotrophs. Particulate matter may be present in the influent, or may consist of products from biomass decay. The products of hydrolysis are phosphate (PO_4-P) soluble organic nitrogen (N_{OS}), and readily biodegradable COD (S_{bsc}).
- Soluble organic nitrogen undergoes ammonification to produce ammonia (NH_3-N). This process is mediated by the heterotrophs (Non-polyP and polyP).
- Precipitation of struvite and calcium phosphates removes PO_4-P and NH_3-N from solution. These are kinetically controlled processes described in the **Spontaneous Chemical Precipitation** section of the "Process Model Formulation" chapter.
- Non-polyP heterotrophs ferment the complex readily degradable COD (S_{bsc}) to acetic acid (S_{bsa}), propionic acid (S_{bsp}) hydrogen (S_{bH2}), and carbon dioxide (S_{CO2t}). There are two model processes for this reaction step. One is for low dissolved hydrogen concentrations while the other is for high dissolved hydrogen concentrations. The stoichiometry of each of these processes can be calibrated to achieve the appropriate product mix (see the **Fermentation** section of the "Process Model Formulation" chapter).
- Dissolved hydrogen (S_{bH2}) and carbon dioxide (S_{CO2t}) are stripped from solution at a rate that is proportional to the difference between the saturated dissolved component concentration and the actual dissolved component concentration (see the **Aeration and Gas Transfer Model** section of the "Process Model Formulation" chapter. It is only the undissociated carbonic acid form (H_2CO_3) of the S_{CO2t} component that is stripped from solution, so carbon dioxide stripping is pH dependent (see the **Modeling of pH in BioWin** chapter).
- Propionic acid (S_{bsp}) is converted to acetic acid by acetogenic bacteria. This process also produces hydrogen and is switched off at high hydrogen concentrations.
- Methane production occurs as a result of the growth of two different groups of organisms. Acetoclastic methanogens consume acetic acid. The substrate of hydrogenotrophic methanogens is dissolved hydrogen and dissolved carbon dioxide.
- The growth of the heterotrophs, acetogens, and methanogens is switched off at high pH and at low pH.



Conceptual schematic for the anaerobic degradation model.

Activated Sludge vs. Anaerobic Digestion

There are some differences in the model processes in activated sludge vs. anaerobic digestion. These are noted here:

- The anaerobic digester hydrolysis process is separate from the activated sludge process. Hydrolysis rate parameters are set specifically for the digester.
- The anaerobic fermentation process is the same for both the anaerobic digester and activated sludge elements. However, the fermentation rate is reduced in the activated sludge process.

There are a number of processes in the model that, although present in the activated sludge process, are not likely to be significant under the typical conditions in that element. These are described below:

- The growth rate of acetogens is very low at ambient temperatures, and these organisms only grow under anaerobic conditions. In addition, the acetogen substrate concentration, S_{bsp}, is typically quite low in an activated sludge process. For these reasons, propionate acetogens will usually washout in the activated sludge process.
- Methanogen growth rates are very low at ambient temperatures, and methanogens are obligate anaerobes. As a result, methanogens will usually washout in the activated sludge process.
- In an anaerobic digester, hydrogen COD (S_{bH2}) is converted to methane. In an activated sludge system, there is an insignificant concentration of hydrogenotrophic methanogens, so that S_{bH2} formed is largely stripped through aeration.
- The struvite precipitation model process exists in both the activated sludge and the anaerobic digester elements. However, struvite precipitation only occurs at the high NH_3-N, PO_4-P and alkalinity concentrations usually found in the anaerobic digester.

Anaerobic Digestion of BNR Biosolids – A Case Study

Case Study Description

To investigate the impact of the anaerobic digester on biological phosphorus removal behavior, data from a municipal BNR demonstration study at the York River, Virginia WWTP were examined (Randall, et al., 1992). A BioWin^{TM}2.1 configuration for the plant during the period it was operated as an anaerobic-aerobic process is shown in **Figure 2**. The BNR configuration includes a mixed liquor recycle, but the recycle rate was set to zero during the period of operation considered in this simulation study. Primary solids and waste activated biosolids were combined and thickened prior to anaerobic digestion. A small bioreactor is included in the simulator configuration following the digester to represent the point where the stripping of carbon dioxide and a subsequent rise in pH of the digested solids would occur. Biosolids are dewatered in the configuration, and in the actual plant, filtrate was returned to the primary influent. Simulation examined the impact of this filtrate by looking at a "No recycle" case in addition to the actual case. A list of the BNR bioreactor process elements and sizes is included in **Table 1.**

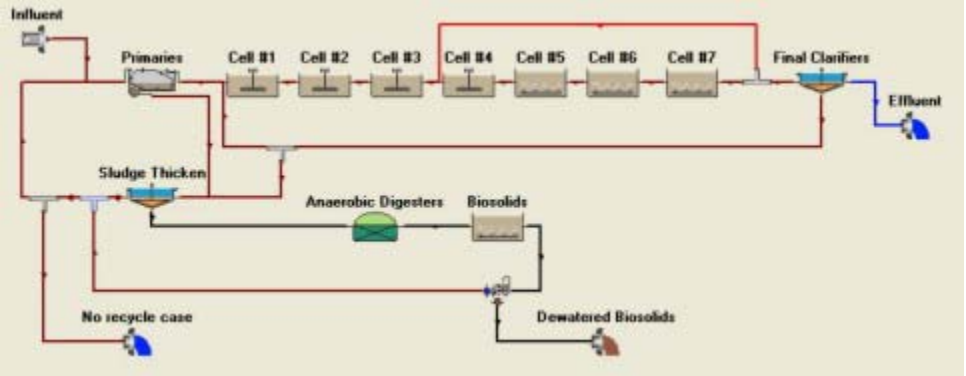


Figure 1 – *BioWin™ 2.1 schematic for the York River WWTP operated as an anaerobic-aerobic process during a BNR demonstration study.*

Unit Process Element	Description	Volume (ML)
Cell #1	Anaerobic Zone	0.42
Cell #2	Anaerobic Zone	0.42
Cell #3	Anaerobic Zone	0.42
Cell #4	Anaerobic Zone	0.42
Cell #5	Aerobic Zone	0.85
Cell #6	Aerobic Zone	1.27
Cell #7	Aerobic Zone	1.27
Total BNR Liquid Volume		5.07

Table 1 - *BNR Bioreactor process elements in the simulated York River BNR Case Study*

The simulation case study considered data from the York River BNR demonstration period from October, 1986 to April, 1987. **Table 2** summarizes the measured influent characteristics during this period.

Parameter	Value	Units
Average Flow	32.9	ML/d
COD	374	mg/L
BOD_5	172	mg/L
TP	8.7	mg/L
TKN	27.9	mg/L

Table 2 - *Measured raw influent characteristics at the York River BNR Demonstration for the period from October, 1986 to April, 1987.*

The BioWin™ 2.1 influent characteristics were calibrated using the available data. The resulting calibrated influent parameters are shown in **Table 3**. Steady state simulations results were compared to reported average plant operating parameters during the period of interest. The results are summarized in **Table 4**. Although there are some differences between the reported measurements and the simulation

results, the values are reasonably close for the purpose of illustrating the impact of solids digestion recycles on the BNR process.

Parameter	Description	Value	Units
f_{bs}	Readily biodegradable	0.14	mg COD/mg total COD
f_{xsp}	Non-colloidal slowly biodegradable	0.75	mg COD/mg slowly biodegradable COD
f_{us}	Unbiodegradable soluble	0.08	mg COD/mg total COD
f_{up}	Unbiodegradable particulate	0.18	mg COD/mg total COD
f_{na}	Ammonia	0.66	mg NH_3-N /mg TKN

Table 3 - *Calibrated BioWin™ 2.1 influent characteristics*

Parameter	Measured (mg/L)	Simulated (mg/L)
Primary Effluent Concentrations:		
COD	255	219
BOD_5	114	106
Filtered BOD_5	45	75
TKN	30	30
TP	10.6	10.6
Bioreactor Concentrations:		
MLSS	2,740	2,800
MLVSS	1,900	1,950
Waste Activated Solids Concentrations		
TSS	6,100	6,200
VSS	4,220	4,330
TP	430	520

Table 4 - *Simulated and actual average operating parameters for the period October, 1986 to April, 1987*

Case Study Results

As an initial indication that the simulation of the digesters was reasonable, the simulated biosolids produced in the case study were used in a series of steady state simulations of the anaerobic digesters to check the dependence of predicted performance on digester hydraulic retention time (HRT). Predicted volatile suspended solids (VSS) destruction is shown in **Figure 3**. In the typical operating region of 20 to 30 days, the VSS destruction is approximately 50%, which is within an expected range of typical performance (Parkin and Owen, 1986). The digesters at York River were reported to be underloaded with an unusually long retention time. In this case study, they were simulated with a 90 day HRT, resulting in a VSS destruction of 60%.

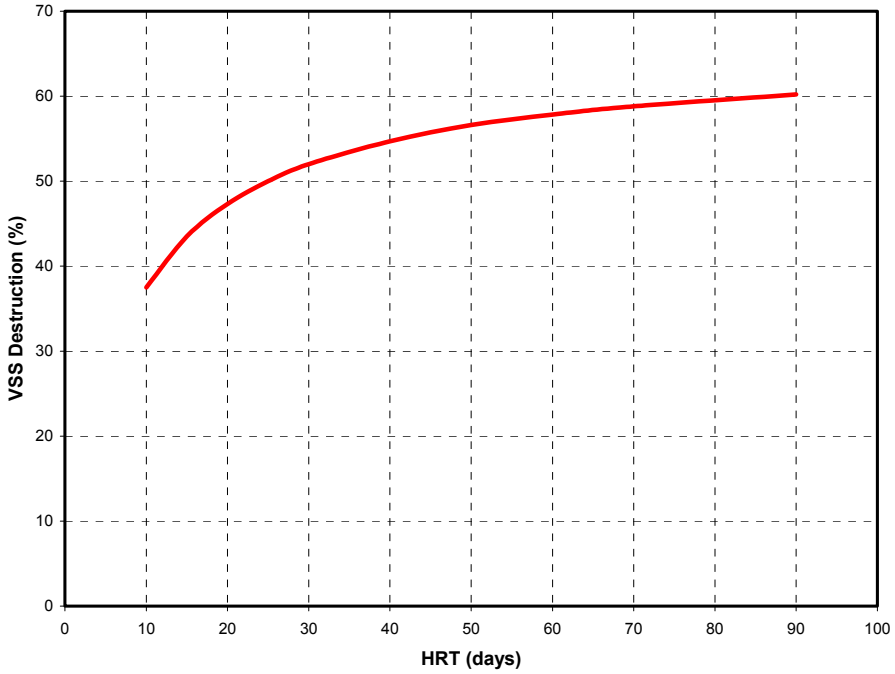


Figure 2 – *Predicted VSS destruction as a function of digester HRT.*

Simulation of Sidestream Impact on BNR Processes

A key concern is the amount of nutrient release in the digesters, and therefore the additional loading on the BNR process caused by the return of solids processing sidestreams to the liquid train. To illustrate this impact, a series of simulations was conducted to look at the amount of digester influent N and P released as a function of digester HRT. Two cases were examined for each nutrient. First, the system was simulated without considering struvite precipitation. Second, the struvite precipitation reactions were included in the simulations. The results are summarized for N as a digester concentration and as a percent of the influent N released in **Figures 4** and **5**, respectively. The corresponding results are shown for P in **Figures 6** and **7**. If struvite precipitation did not occur, more than 50% of the nitrogen in the solids would be released at a 90 day HRT, resulting in an ammonia-N concentration of about 500 mg/L in a filtrate recycle. Struvite precipitation would reduce the net release of nitrogen to about 35%, resulting in a recycle ammonia-N concentration of about 350 mg/L. The proportion of phosphorus released in the digester is even greater due to the storage of excess phosphorus in the biological phosphorus removal process. If struvite precipitation did not occur, more then 70% of the phosphorus in the digester influent solids would be released at a 90 day HRT, resulting in a PO_4-P concentration of more than 600 mg/L in a filtrate recyle. When struvite precipitation reactions are considered, the proportion of P released is reduced to about 50%, and the corresponding PO_4-P concentration in a filtrate recycle stream would be reduced to about 400 mg/L.

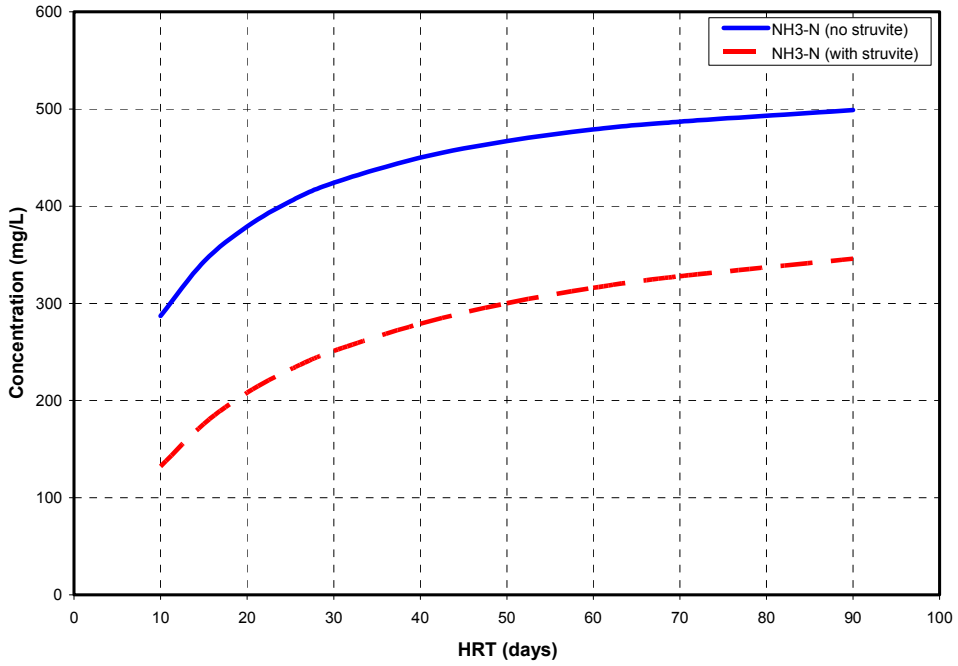


Figure 3 – *Predicted digester NH_3-N concentrations as a function of digester HRT for digester simulation with and without struvite precipitation reactions.*

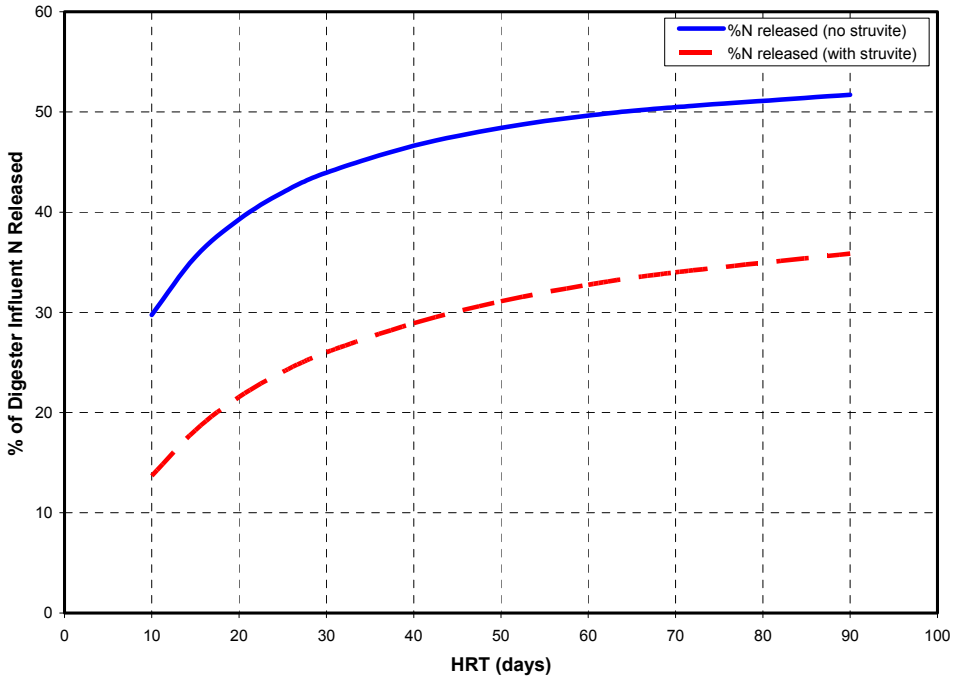


Figure 4 – *Predicted percent of digester influent N released as a function of digester HRT for digester simulation with and without struvite precipitation reactions.*

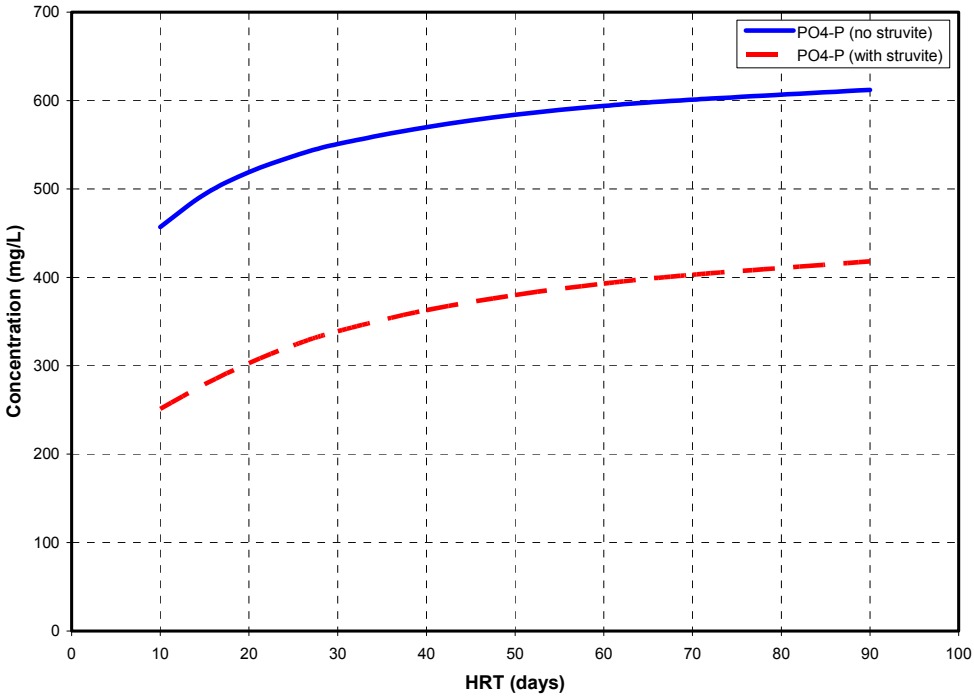


Figure 5 – *Predicted digester PO_4-P concentrations as a function of digester HRT for digester simulation with and without struvite precipitation reactions.*

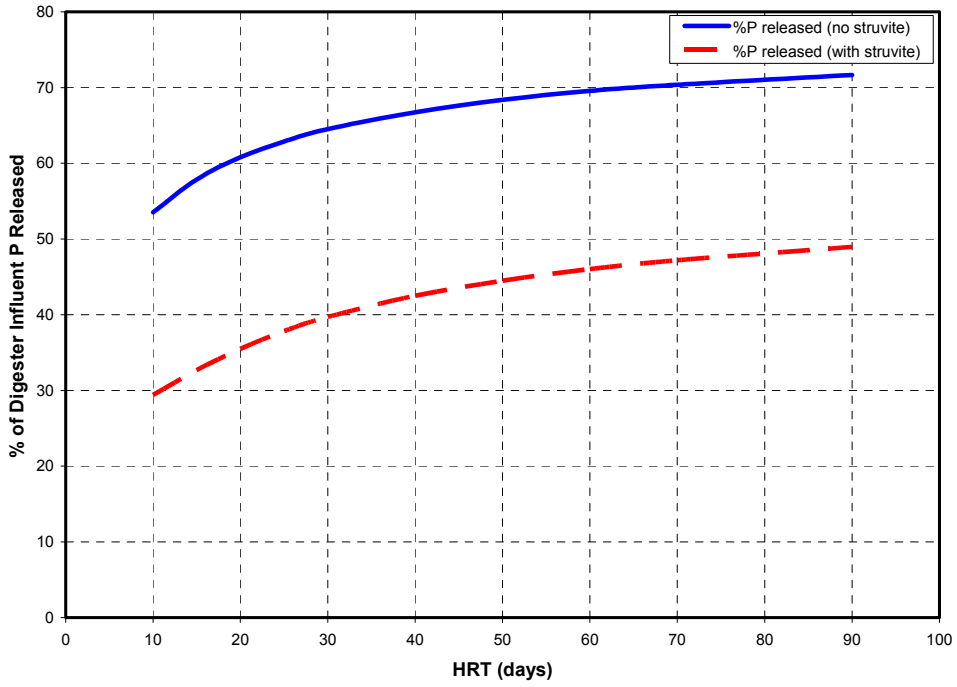


Figure 6 – *Predicted percent of digester influent P released as a function of digester HRT for digester simulation with and without struvite precipitation reactions.*

To illustrate the importance of digester simulation in analysing the overall behaviour of the BNR plant, the impact of the PO_4-P in the recycle streams was examined by comparing simulated primary effluent and final effluent TP concentrations to reported results for three cases:

- No recycle of belt press filtrate;
- Recycle of the filtrate but without considering struvite precipitation in the digester;
- Recycle of the filtrate with struvite precipitation reactions included in the digester simulation.

The simulated results for each of the cases are compared to reported results in **Figure 8**. At the plant, the primary effluent TP concentration typically was about 2 mg/L greater than the raw influent concentration. The simulated results confirm that this increase was due to the belt press filtrate recycle. The primary effluent without the recycle would have decreased to less than 6 mg/L, resulting in a final effluent concentration of less then 1 mg/L. If the recycle concentrations predicted without considering precipitation reactions in the digester were included in the simulation, the resulting primary effluent concentration would be 12.5 mg/L, resulting in a final effluent concentration of 5.6 mg/L. The predicted primary effluent and final effluent TP concentrations were very close to the reported values when precipitation reactions were included in the digester simulation.

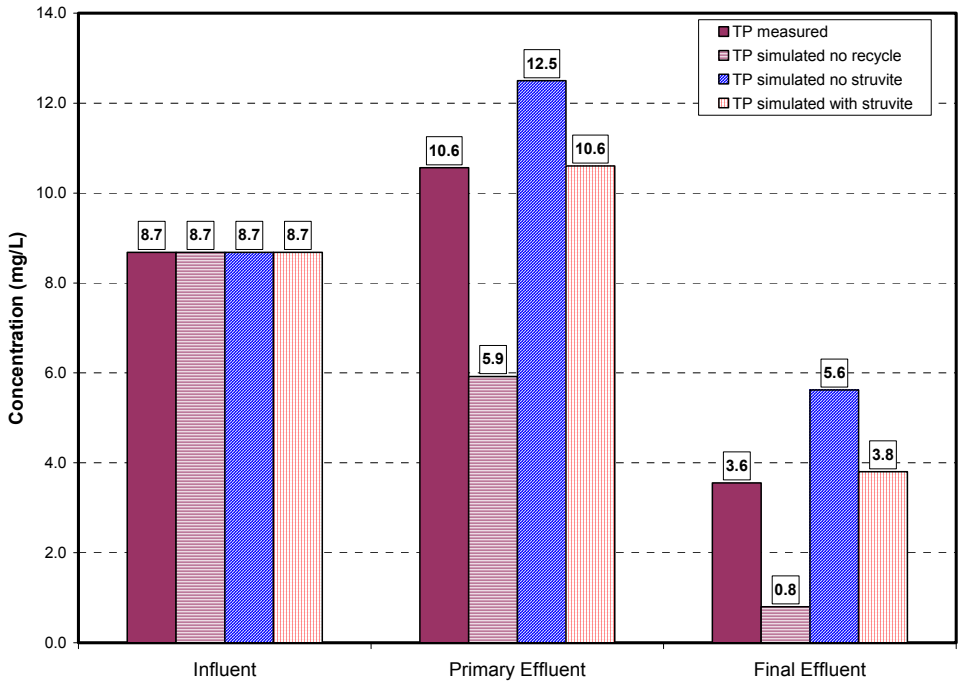


Figure 7 – *Measured and predicted raw influent, primary effluent and final effluent TP concentrations for three cases of digester simulation; no recycle of belt press filtrate; recycle when the digester is simulated without struvite precipitation; recycle when struvite precipitation reactions are included in digester simulation.*

References

Andrews, J. F. (1969). Dynamic model of the anaerobic digestion process. J. Sanit. Engng. Div. Proc. ASCE, 95 (SA1), pp. 95-116.

Andrews, J. F. and Graef, S. P. (1971). Dynamic modeling and simulation of the anaerobic digestion process. Anaerobic Biological Treatment Processes. Advances in Chemistry Series, R.F. Gould (Ed.), 105, American Chemical Society, New York, pp. 126-162.

Bagley, D. M. and Brodkorb, T. S. (1999). Modeling microbial kinetics in an anaerobic sequencing batch reactor – Model development and experimental validation. Water Environment Research, 71 (7), pp. 1320-1332.

Costello, D. J., Greenfield, P.F., Lee, P.L. (1991). Dynamic modeling of a single stage high rate anaerobic reactor – I. Model derivation. Wat. Res., 25, pp. 847-858.

De Souza Araújo, L., Catunda, P.F.C., van Haandel, A.C. (1998). Biological sludge stablisation Part 2: Influence of the compostition of waste activated sludge on anaerobic stabilization. Water SA, 24, pp. 231-236.

Hill, D.T. and Barth, C.L. (1977). A dynamic model for simulation of animal waste digestion. J. Wat. Pollut. Control Fed., 10, pp. 2129-2143.

Jones, R.M., MacGregor, J.F., Murphy, K.L., and Hall, E.R. (1992). Towards a useful dynamic model of the anaerobic digestion process. Wat. Sci. Technol., 25 (7), pp. 61-72.

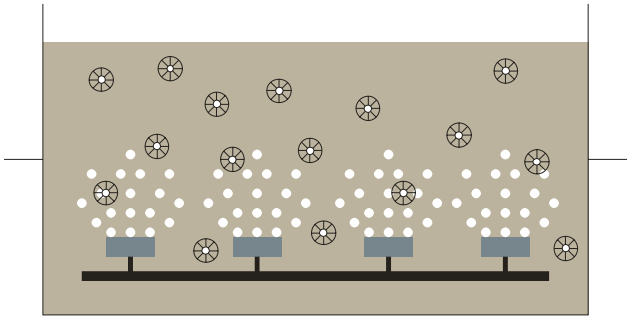
Massé, D.I., and Droste, R.L. (2000). Comprehensive model of anaerobic digestion of swine manure slurry in a sequencing batch reactor. Wat. Res., 34, pp. 3087-3106.

Mosey, F.E. (1983). Mathematical modeling of the anaerobic digestion process: regulatory mechanisms for the formation of short chain volatile acids from glucose. Wat. Sci. Tech., 21, pp. 187-196.

Biofilm

Modeling Fixed Film Processes In BioWin

The objective of this chapter is to describe the biofilm model provided in BioWin. The model is implemented in the media bioreactor element and is calibrated to simulate MBBR and IFAS systems.



The introduction provides general background information on the model.

Model Formulation : this section describes the model structure, including the underlying assumptions. The model belongs in the 1D dynamic layered biofilm model category, with modifications that allow it to be used with one parameter set for a large range of process situations. The biofilm model is integrated with a general Activated Sludge/Anaerobic Digestion model combined with a chemical equilibrium, precipitation and pH module. This allows the model to simulate the complex interactions that occur in the aerobic, anoxic and anaerobic layers of the biofilm.

Model Calibration : this section describes the basic process performance of the model with the default parameter set.

Pre-Configured File Cabinet *: Click the arrow next to this button (at the top of the main window) to select and load pre-configured BioWin process files.*

Examples : this section contains several simulation examples. The model is shown to match a variety of design guidelines, as well as experimental results from batch testing and full-scale plant operation. Both Moving Bed BioReactors (MBBR) and Integrated Fixed Film Activated Sludge (IFAS) systems were simulated using the same model and parameter set. Several examples are provided for the user in the Pre-Configured File Cabinet (found on the BioWin main window toolbar).

Introduction

Several biofilm models have been published over the past 20 years. These vary in complexity from simple analytical models to full 3-D dynamic models. An extensive summary is provided by Wanner et al. (2006). The simpler models are easy to solve, but usually do not capture enough details of the important engineering considerations. Their parameters are highly variable from situation to situation, and design engineers often cannot allocate the time and effort required to calibrate them for different processes and process conditions. On the other hand, the complex models, despite their sound fundamentals, are still not used widely in engineering practice, unlike the activated sludge process models. The reason seems to be that complex, detailed models require too much computer time to solve, and are impractical for everyday engineering use. Their major purpose is for research.

For the specific purpose of engineering design and analysis, a balance is required between the simplified and the complex mechanistic approach. This model should be based on biofilm and diffusion theories but include certain empirical features that ensure solution stability and reasonable computational times. The biofilm model must be integrated with a biological process model which includes all relevant reactions occurring in current wastewater treatment systems. Such a model should see widespread application in engineering practice. This in turn will provide better designs and a basis for improving the model itself. The engineering objective is to have a model that requires minimal calibration, and works for a wide range of loadings and configurations, such as IFAS, MBBR, trickling filter (TF), rotating biological contactor (RBC), biological aerated filter (BAF) with the same parameter set. In BioWin 3.0, the physical setup for IFAS and MBBR systems is provided. However, other fixed film processes may be approximated using the existing model.

Model usage

The BioWin user working with wastewater treatment plants employing fixed film technology will be able to obtain answers for the following questions:

- How much carrier material (i.e. area for biofilm growth) is required to achieve a specific process objective?
- How much active biomass will the biofilm contain?
- How thick will the biofilm grow (in mm and in $gTSS/m^2$) depending on loading and turbulence conditions?
- How will soluble and particulate components be distributed inside the biofilm?
- What part of the biofilm will be penetrated by DO, substrate, what will be anoxic, and what part will be anaerobic?
- How will the reaction rates for the biological processes (growth and decay of heterotrophic and autotrophic biomass, fermentation, etc.) be distributed within the biofilm?
- How much will release of gases (N_2, CH_4, CO_2) in the deeper layers of the biofilm contribute to solids detachment?
- Will pH effects and precipitation reactions within the biofilm significantly change porosity, density and conversion rates?

The typical use of models in BioWin is design, optimization, training and operations. This requires simplicity, stability, quick solution times and proper process

performance predictions on a wide range of systems. The BioWin biofilm model achieves these objectives through the use of:

- a well calibrated, advanced biological/chemical model, including aerobic, anoxic, anaerobic, fermentation, pH, chemical equilibrium, precipitation and gas transfer processes;
- a one dimensional (1D) fully dynamic biofilm model configuration;
- modified attachment/detachment and density calculation methods;
- one validated parameter set for a wide range of systems;
- calibration of the model to various design guidelines, batch test experiments and pilot and full-scale performance indicators on several systems;
- a fast numerical method that provides solutions within a few minutes (note: the BioWin biofilm model is a complex model and runs slower than a bioreactor model. However every effort was made to provide a reasonable convergence and dynamic runtime).

Model Formulation

The dynamic mixed-culture biofilm model implemented in BioWin belongs to the class of 1D models as described by Wanner and Reichert (1996) and Reichert and Wanner (1997). 1D models describe soluble and particulate component profiles perpendicular to the media. Fundamental equations are listed in the two references. The model includes diffusion of soluble and particulate components, a boundary layer between the biofilm surface as a diffusion resistance for solutes, exchange of particulates due to detachment (erosion) and attachment (impingement) of solids between the film surface and the bulk liquid concentration. Film thickness can change dynamically due to substrate loading and solids exchange between the film and the bulk liquid.

There are several differences and additions to the Wanner/Reichert model to improve its ability to predict typical process conditions using one parameter set in the majority of cases.

The Biofilm Model Uses the Full ASDM Process Model

Due to the complexity of the numerical solution, 1D biofilm models have typically been integrated with simple biological models. For example, models with 5-10 components [one or two biomasses (X_H, X_A) and two or three limiting substrates (DO, S_S, NH_3), etc.]. The process model integrated with the biofilm model in BioWin is the full General Activated Sludge/Anaerobic Digestion Model (ASDM). The process model integrated with the biofilm model in BioWin is the full General Activated Sludge/Anaerobic Digestion Model (ASDM). This model is described in detail in the Process Model Fornulation Chapter. The model includes reactions for activated sludge and anaerobic digestion environments, as well as pH, gas transfer and chemical precipitation within the same model matrix. The model tracks over 50 components, with more than 80 processes acting on these components. The main processes are summarized below:

- Aerobic heterotrophic growth using complex substrate, acetate, propionate and methanol
- Anoxic heterotrophic growth on NO_3^- and NO_2^- using complex substrate and VFAs

- Anaerobic fermentation of complex substrate, propionate and methanol
- Growth of bio-P microorganisms and storage of polyphosphate
- Various hydrolysis, ammonification and colloid flocculation reactions
- Assimilative nitrate and nitrite reduction
- Anoxic growth of methylotrophs on nitrate and nitrite
- Growth of ammonia and nitrite oxidizer biomasses
- Growth of Anammox microorganisms
- Growth of autotrophic and heterotrophic methanogens
- Decay of all nine (9) active biomasses in different environments
- pH estimation based on the phosphate, carbonate, ammonia, acetate and propionate systems, including strong acids and bases, plus other relevant reactions
- Precipitation of various calcium, magnesium, aluminum and iron complexes (struvite, HDP, HAP, etc)
- Gas transfer of O_2, CO_2, N_2, NH_3, H_2 and CH_4 gases
- Inorganic suspended solids fixation during polyphosphate storage and heterotrophic growth

This complex model has been applied extensively for modelling the large number of processes occurring in suspended growth biological systems, and hence calibration requirements are significantly reduced.

Turbulence and Diffusion

The mean velocity gradient (G-value) approximating turbulence in the bulk liquid is calculated according to Grady *et al*. (1999). Soluble component diffusion is described by Fick's Second Law, with a diffusion resistance added due to the laminar layer at the biofilm surface. Diffusion coefficients are taken at 80% of clean water values. The model provides a simple "streamer" function to increase diffusion (through increasing available area) to the top layers of the biofilm in the case of porous films or films with streamers in high turbulence situations (e.g. an aerated IFAS tank).

Attachment/Detachment

Particulate attachment and detachment rates have a major role in establishing biofilm thickness, dynamics, and system activity, similar to SRT in activated sludge. Particulate attachment in this model is related to bulk particulate concentration. Detachment is a combined function reflecting the most important variables affecting film detachment, such as film thickness or mass per area in the film, and the effect of N_2 or CH_4 gas generation inside deeper film layers. Turbulence (G-value) is not included explicitly in the attachment and detachment expressions above. An environment with a higher G-value will usually result in a denser, more resistant biofilm, not necessarily a thinner film.

Biofilm Density

There are various mathematical descriptions in the literature to approximate density and porosity (Wanner *et al*., 2006). In the BioWin model the long term activity is

linked to overall biofilm density through the abundance of individual particulate state variables. Each particulate state variable has a characteristic density or maximum concentration in the film. For example, heterotrophs and autotrophs grow to a maximum concentration of 50 and 100 kgCOD.m^{-3}, respectively, while inorganic solids have a maximum density of 1300 kg.m^{-3} in the film. Particulate substrate (X_S) and organic inerts (X_I) are assigned a low density (5 kgCOD.m^{-3}). The total solids density, in each layer, is a weighted combination of the individual particulate variable densities and is used to estimate biofilm volume, thickness and the density profile. Inorganic suspended solids are less "sticky" and will detach easier than organic variables. This accounts for higher VSS/TSS ratios found in biofilms.

Biofilm parameters at a glance

Project|Parameters|Biofilm

Name	Default Value	Typical use
Attachment rate [g / (m^2 d)]	80	
Attachment TSS half sat. [mg/L]	100	
Detachment rate [1/d]	8.00E+04	Increase to reduce biofilm thickness
Solids movement factor [-]	10	
Diffusion neta [-]	0.8	Effects all diffusivities (min 0.5)
Thin film limit [mm]	0.5	
Thick film limit [mm]	3	
Assumed film thickness for tank volume correction [mm]	0.75	
Film surface area to media area ratio - Max.[ ]	1	Increase to form streamers (max 1.5)
Minimum biofilm conc. for streamer formation [gTSS/m^2]	4	

Project|Parameters|State variable properties

This menu item provides access to three groups of parameters:

- maximum biofilm concentrations
- Effective diffusivities
- EPS strength coefficients

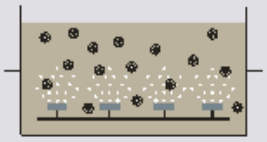
Flyby panes

Flyby panes report the following values for the Media Bioreactor (example is using a typical IFAS reactor):

Physical parameters (bottom left):

Name: Media Reactor East	Type: Media Bioreactor	
Volume:	179.00	m3
Area:	52.65	m2
Depth:	3.40	m
Diffuser coverage:	1.00	%
Number of diffusers:	13	
Diffuser unit area:	0.0410	m2
Media area	15863.52	m2
Reactor media fill	22.90	%
Media specific area	387	m2/m3
Media specific volume	0.125	m3/m3
Media displaced volume	5.12	m3

Performance parameters (bottom right):



		Hydraulic residence time	
		Film bottom	top
Ammonia N		4.05	4.04
Nitrate N		11.85	11.83
Nitrite N		1.54	1.54
Soluble PO4-P		2.87	2.87
Film Thickness	1.19 mm		
TSS / Area	16.25 g / m2		
Displaced Volume (biofilm + media)	24.03 m3		
Total oxygen uptake rate	14.01 mgO/L/hr		
Volatile suspended solids		5609	7725
Total suspended solids		12000	16425
Dissolved oxygen		0.01	0.41
Actual DO sat. conc.	10.24 mg/L		
Air flow rate / diffuser	39.61 m3/hr (20C, 101.325 kPa or 1 atm)		

pH

Power

Mech. mixing

Air / flow

Velocity gradient

Model Calibration

Calibration to Design and Engineering Criteria

Various design loading guidelines for IFAS and MBBR systems were checked against model performance for each loading condition. A range of characteristics were monitored: DO and substrate penetration, heterotrophic and autotrophic activity and location, the level of nitrification, substrate removal efficiency, film thickness, biofilm mass per unit area, denitrification, fermentation, gas production in deep biofilm layers, as well as pH. The model provides results according to engineering expectations. A few selected examples are shown in Table 1.

Table 1. *Calibration to design guidelines*

System loading level		Low	Medium	High
MBBR :				
Design BOD loading	$gBOD.m^{-2}.d^{-1}$	2.0	6.7	15.0
Model film thickness	mm	0.1	0.5	1.0
Model mass	$gTSS.m^{-2}$	1.5	11.0	14.7

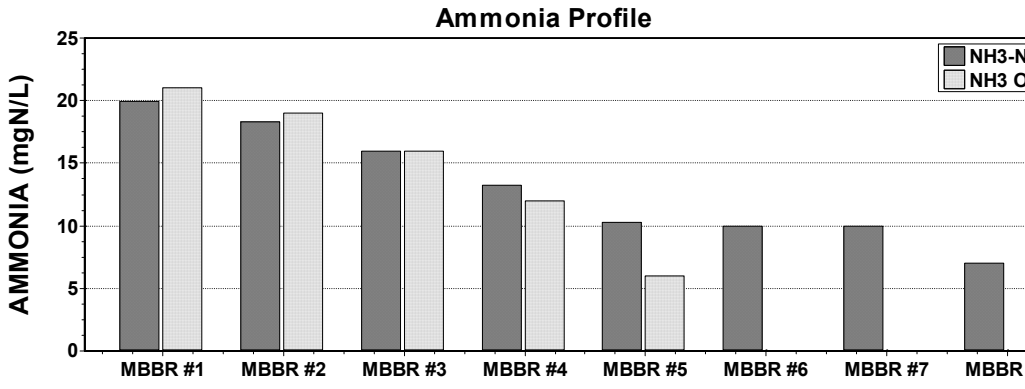
Model effluent NO_3	$gN.m^{-3}$	14.5	0	0
IFAS :				
Design BOD loading	$gBOD.m^{-2}.d^{-1}$	3.3	10.0	16.7
Model film thickness	mm	0.7	0.9	1.7
Model mass	$gTSS.m^{-2}$	8.3	14.2	22.1
Model effluent NO_3	$gN.m^{-3}$	18.6	19.7	15.9

Examples

MBBR System

Rusten *et al*. (1994) conducted an MBBR demonstration study on one of a number of parallel trains at the 90,000 $m^3.d^{-1}$ Bekkelaget treatment plant. The MBBR unit received an average primary effluent flow of 5200 $m^3.d^{-1}$. The process consisted of eight reactors in series with a total volume of 568 m^3 (average empty bed HRT of 2.6 hours). Carrier elements provided an active biofilm surface area of approximately 300 $m^2.m^{-3}$ in each zone. The first five reactors were aerated for removal of organic material and nitrification. Methanol for denitrification was introduced into Reactor 6, and Reactors 6 and 7 were unaerated. Reactor 8 provided post-aeration.

Rusten *et al*. (1994) provide only limited information on (a) influent concentrations and wastewater characteristics; (b) influent load and temperature variations with time during the study; and (c) bulk DO concentrations in aerated zones. Therefore it is not possible to conduct a comprehensive modelling exercise. However, the paper does show ammonia and nitrate concentration profiles in the first 5 aerated zones for typical 'high' and 'normal' load situations. The 'normal' load case was simulated to assess whether the model would at least predict reasonable trends in ammonia and nitrate. The results are shown in Figure 1, comparing the model predictions (darker shaded bars) to the reported data (lighter shaded bars). The simulation shows reasonable correspondence between predicted and observed profiles. The under-prediction of nitrate in Reactor 5 (and ammonia over-prediction) could be corrected readily by modifying bulk DO concentration. However, the purpose is not to force a fit to the data given the uncertainties over operating conditions; rather, the purpose is to demonstrate that the model predicts the general performance reasonably.



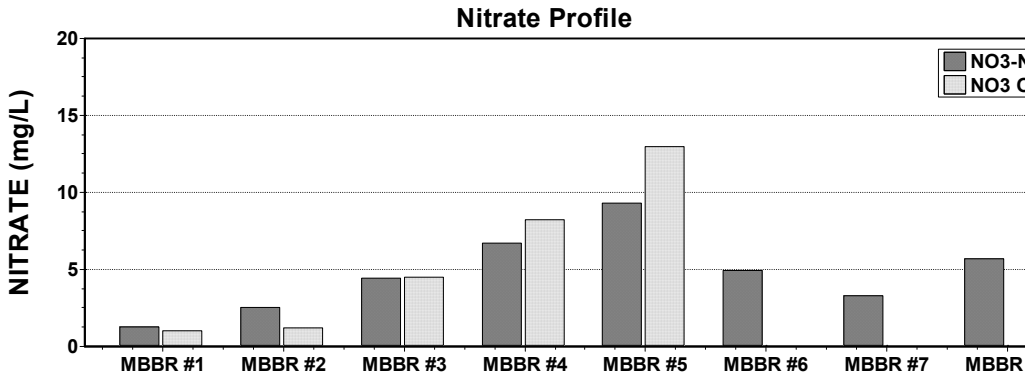


Figure 1: *Approximate simulation of ammonia and nitrate profiles in the Bekkelaget MBBR demonstration study of Rusten et al. (1994). [Darker bars are model predictions; lighter bars are reported data for Reactors 1 to 5].*

IFAS System – Simulation of Full-Scale Performance Demonstration

The Waterdown STP is an activated sludge plant in Ontario, Canada with a design capacity is 2700 $m^3.d^{-1}$. Between October 1995 and June 1998, studies were conducted at Waterdown to evaluate IFAS options for reducing the cost of retrofitting larger plants for year-round nitrification (Jones *et al*., 1998; Jones *et al*., 1999). Between April 1997 and June 1998, an IFAS process that employed a plastic free-floating media was evaluated. The study was conducted in a section of the plant (Plant A) that consists of two separate trains each with 224 m^3 aeration tanks in parallel. For the IFAS studies, media was introduced in one tank with the other train serving as the control. In each aeration tank a baffle and a mixer were installed to create a separate 44 m^3 anoxic/aerobic swing zone (20% volume fraction) at the influent end of the tank. Primary effluent was split equally between the two trains, and each train had separate wasting (WAS) from the mixed liquor, separate secondary clarification and separate return activated sludge (RAS).

The IFAS study used media that consisted of polypropylene plastic cylinders that were 22 mm in diameter and approximately 15 mm long with a specific (internal) surface area of 389 $m^2.m^{-3}$ at a 100% fill volume. On April 11, 1997, 24 m^3 of the media was installed in the section of the aeration tank downstream of the baffle in one of the aeration trains. On January 20, 1998, an additional 17 m^3 of media was added to the east aeration tank, resulting in a total fill fraction of 18 % based on the total aeration tank volume (22% in media zone). The performance of the media and control trains was monitored over the period from December, 1997 until June 1998 to span the winter months. The process temperature during this period reached a minimum of 8^oC. The solids retention time was controlled to the same target of 3 days in both trains in an attempt to stress the nitrification performance.

The model configuration of the Waterdown STP media and control trains is shown in Figure 2. The December, 1997 to June, 1998 period was simulated by inputting daily primary effluent flows and wastewater characteristics, process temperatures, WAS flows and RAS flows, and running a dynamic simulation for the study period. Simulated and measured values of MLSS in each train are shown in Figure 3. The excellent agreement of the simulated and measured values confirms the loading and wastage rates on the two trains. Figure 4 shows the media and control train effluent ammonia (NH_3-N) concentrations. The figure demonstrates that the model was able to correctly predict the enhancement in nitrification performance provided by the media.

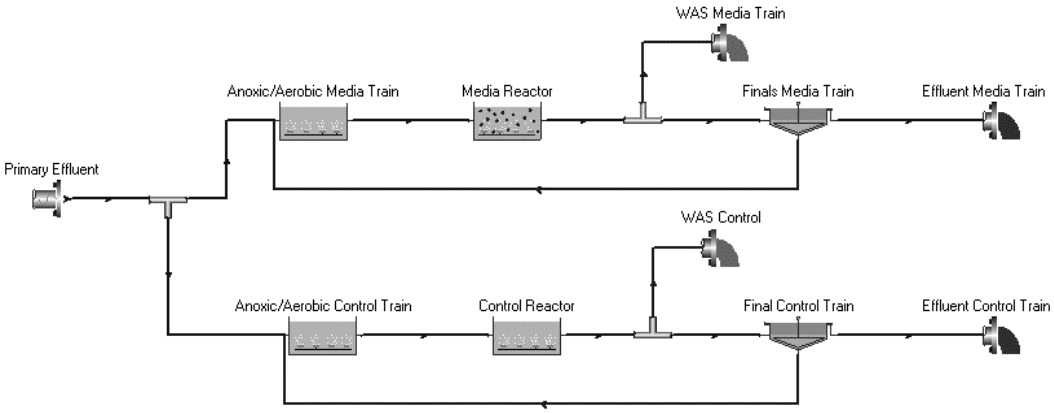


Figure 2: *Configuration for the Waterdown STP Media and Control trains.*

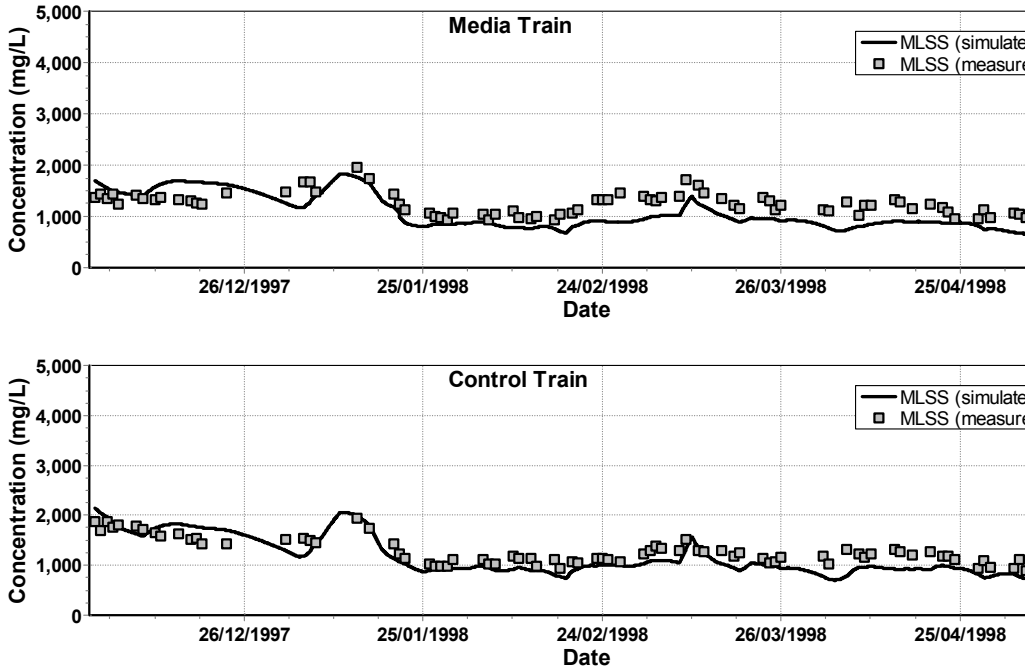


Figure 3: *Simulation of MLSS concentrations in the media and control trains.*

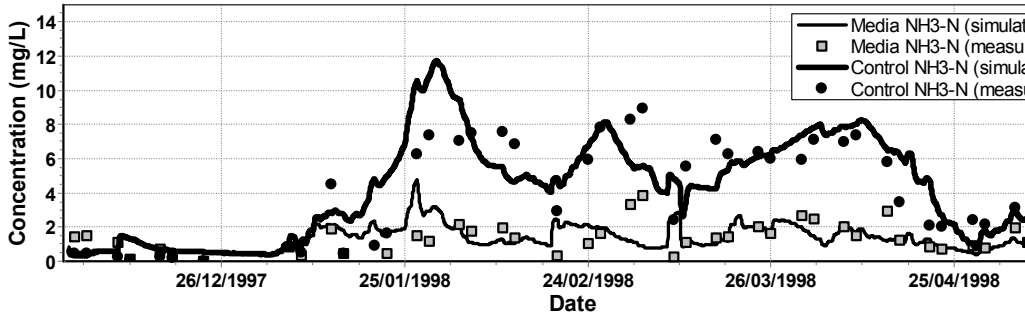


Figure 4: *Simulation of effluent NH3-N concentrations in the media and control trains.*

IFAS System - Aerobic Batch Test

Yerrell *et al*. (2001) reported on a full-scale demonstration of the IFAS system using a free-floating plastic media at a 7500 $m^3.d^{-1}$ module of the Christies Beach WWTP in South Australia. The purpose of the project was to evaluate whether the existing plant could be retrofit to attain complete nitrification and partial denitrification without adding bioreactor volume or settling capacity. The plant was step-fed with primary effluent and contained two anoxic zones (no media), and three IFAS zones each with an active biofilm surface area of approximately 86 $m^2.m^{-3}$.

During the study batch testing of mixed liquor alone and mixed liquor with IFAS media removed from the aeration tank allowed an assessment of the nitrification enhancement. The initial ammonia concentration was increased to 20 or 30 $gN.m^{-3}$ by adding NH_4Cl at the start of the test, and then supplemented to maintain an ammonia concentration greater than 15 $gN.m^{-3}$ during the batch test. The appropriate amount of alkalinity to maintain a stable pH of approximately 7.2 was supplied by

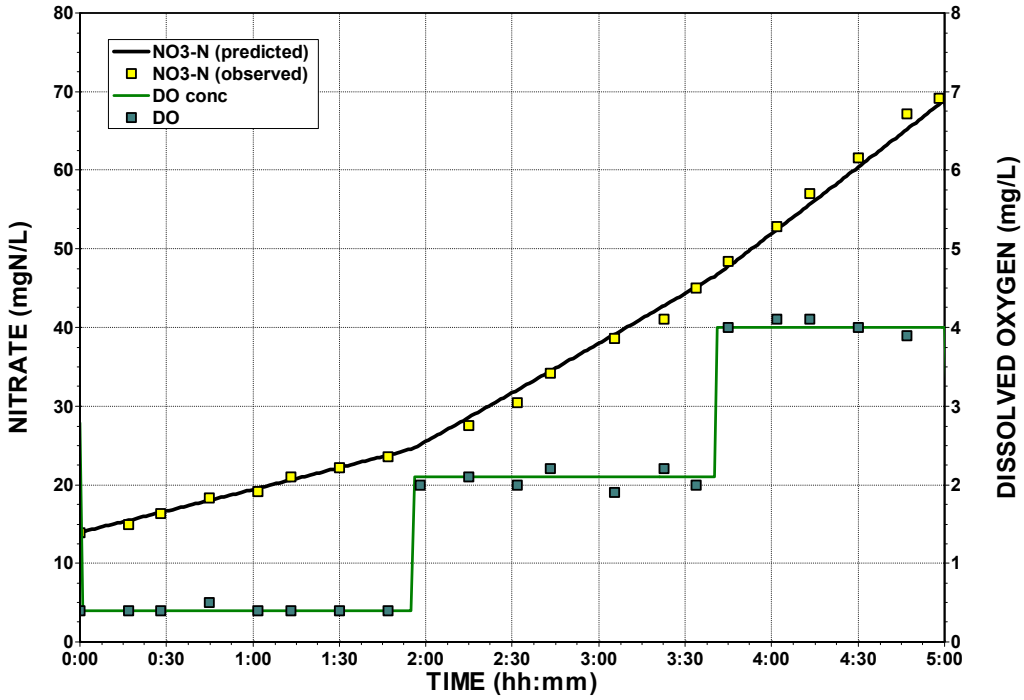


Figure 5: *Predicted and observed nitrate response in the IFAS batch test with dissolved oxygen (DO) concentration increasing in stages from 0.5 to 2.0 and 4.0 $g.m^{-3}$.*

adding $NaHCO_3$. Figure 5 shows an example of a batch test (40 L volume) conducted with mixed liquor (and media) removed from the IFAS-3 reactor. Aeration was adjusted during the test to step the bulk DO concentration from 0.5 to 2.0 and to 4.0 $g.m^{-3}$ at intervals of approximately 2 hours. The nitrate response shows three distinct linear phases at each DO concentration. The figure shows the results of the simulated batch test response as well. The model predicts the increasing nitrate production rate with each increase in DO concentration.

References

Grady, C.P.L., Daigger, G.T. and Lim, H.C. (1999). Biological Wastewater Treatment. Marcel Dekker, New York.

Jones, R.M., Sen, D. and Lambert, R. (1998). Full scale evaluation of nitrification performance in an integrated fixed film activated sludge process. Wat. Sci. Tech. 38 (1), 71-78.

Jones, R., Lambert, R., Manoharan, R., Crane, R. and Campbell, H. (1999). Demonstration Of An Integrated Fixed Film Activated Sludge (IFAS) System For Enhanced Nitrification Using A New Free Moving Media. Proc. of 28th Annual Technical Symposium of the Water Environment Association of Ontario, Toronto, April.

Reichert, P. and Wanner, O. (1997). Movement of Solids in Biofilms: Significance of Liquid Phase Transport. Wat. Sci. Tech. 36 (1), 321-328.

Rusten, B., Siljudalen, J.G. and Nordeidet, B. (1994). Upgrading to nitrogen removal with the KMT moving bed biofilm process. Wat. Sci. Tech. 29 (12), 185-195.

I. Takács, C.M. Bye, K. Chapman, P.L. Dold, P.M. Fairlamb, R.M. Jones: A Biofilm Model For Engineering Design. Wat. Sci. Tech. in press.

Wanner, O. and Reichert, P. (1996) Mathematical Modeling of Mixed-Culture Biofilms. Biotechnology and Bioengineering 49, 172-184.

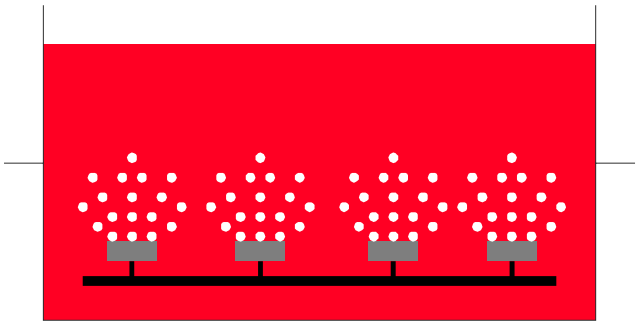
Wanner, O., Eberl, H., Morgenroth, E., Noguera, D., Picioreanu, C., Rittman, B., and van Loosdrecht, M. (2006). Mathematical Modeling of Biofilms. IWA Scientific and Technical Reports #18, IWA Publishing.

Yerrell K, Gobbie, M. Dold, P. Jones R. and Sickerdick, L. (2001). Full-scale demonstration of a free-moving media IFAS process for enhancing nitrification performance. Proc. 74th Annual Conference of the Water Environment Federation, Atlanta (October)

Sidestream Treatment Processes

Modeling Sidestream Treatment Processes in BioWin

The objective of this chapter is to describe how sidestream treatment processes are modeled in BioWin. A sidestream reactor element is available to represent these processes in BioWin configurations.



The Introduction provides general background information on modeling sidestream processes.

Model Formulation : this section describes the model structure. Sidestream modeling employs the general Activated Sludge/Anaerobic Digestion model combined with a chemical equilibrium, precipitation and pH module. This allows the model to simulate the complex interactions that occur in sidestream processes.

Model Calibration : this section describes a number of cases where the model performance was compared to various process systems where suitable data was available for calibration.

Examples: this section contains several simulation examples. Several examples are provided for the user in the Pre-configured File Cabinet.

Pre-Configured File Cabinet *: Click the arrow next to this button (at the top of the main window) to select and load pre-configured BioWin process files.*

Introduction

Anaerobic digestion of waste activated sludge results in release of nitrogen as ammonia. Typically digested sludge is dewatered before disposal, and the liquid stream from the dewatering step is returned to the activated sludge process. The nutrient load of the reject water stream is considerable, and can increase the influent nitrogen load by 15-25%. This additional load increases the cost and complexity of meeting stringent effluent requirements for total nitrogen (TN). Also, the capacity of the liquid train to treat additional ammonia load may be limited by factors such as influent alkalinity, insufficient aeration capacity, or insufficient SRT.

A number of 'named' side-stream biological processes have been developed for treating the ammonia component of the reject water before returning it to the liquid train (InNitri, BABE, SHARON, ANAMMOX, CANON, DEMON, etc.). These systems involve one or more of the following biological transformations, or a combination of these:

- Nitritation mediated by ammonia oxidizing autotrophic bacteria – AOBs (i.e. conversion of ammonia to NO2) or partial nitritation (i.e. converting a portion of the ammonia to NO2);
- Nitratation mediated by nitrite oxidizing autotrophic bacteria – NOBs (i.e. conversion of nitrite to nitrate –NO3);
- Denitritation mediated by heterotrophic bacteria where nitrite serves as an electron acceptor on the addition of organic substrate with production of nitrogen gas;
- Denitratation mediated by heterotrophic bacteria where nitrate serves as an electron acceptor on the addition of organic substrate with production of nitrite;
- Nitrogen removal by autotrophic anammox bacteria (ANaerobic AMMonia OXidation). The process converts ammonia directly into nitrogen gas in unaerated conditions, utilizing nitrite as an electron acceptor.

The following terminology is used to describe combinations of the basic biological transformations:

- Nitrification: nitritation followed by nitratation (i.e. conversion of ammonia to nitrite and then nitrate);
- Denitrification: denitratation followed by denitritation (i.e. conversion of nitrate to nitrite and then nitrogen gas);
- Deammonification: partial nitritation (i.e. converting a portion of the ammonia to nitrite) followed by the anammox reaction (i.e. conversion of ammonia and nitrite to nitrogen gas);

A range of benefits have been identified for the different side-stream treatment systems; for example:

- Seeding the activated sludge train with AOBs and NOBs grown in the side-stream stage, allowing shorter SRTs (bioaugmentation).
- Less carbon substrate is required to denitrify nitrite rather than nitrate.
- Less aeration (and alkalinity) is required to convert ammonia to nitrite rather than nitrate.
- In the ANAMMOX process no organic carbon is added for denitrification, and so there is not increased biosolids production or emission of CO2.

Side-stream systems focused on ammonia treatment have been implemented in a number of reactor configurations; for example, single and series flow-through CSTRs, reactor-clarifiers with sludge recycle, SBRs, attached growth systems. The operating conditions of side-stream biological processes are considerably different from those in the main stream process. This leads to a number of unique considerations for operation and control:

- Stopping nitrification at the nitrite stage and preventing nitrate formation relies on the difference in growth rates between AOBs and NOBs, and the different temperature dependencies.
- High concentrations of substrate and product species such as ammonia and nitrous acid can lead to inhibitory conditions for AOBs and NOBs. In some cases successful performance depends on inhibition of certain reaction steps.
- Careful pH control often is required for successful operation.
- Anammox organisms have very low growth rates necessitating long SRTs, and long process start-up times.

Model Formulation

Table 1 summarizes key process considerations that a model for side-stream processes should include. Several models for a number of side-stream processes have been reported. For example, Volcke (2006) developed a two-step nitrification and denitrification model to represent the SHARON process. Wett and Rauch (2003) developed a two-step nitritation-denitritation model based on detailed data from two full-scale reject water SBR treatment processes. Van Hulle (2005) incorporated Anammox reactions into a two-step nitrification/denitrification model.

Table 1: *Summary of key process aspects that a model for side-stream processes must include.*

Process Aspect	Model Process	Important Considerations
Nitrification	• AOB growth and decay • NOB growth and decay	Different growth rates, temperature dependencies and inhibition effects.
Heterotrophic Denitrification	• Growth on substrate through denitritation (using nitrite as an electron acceptor) • Growth on substrate through denitratation (using nitrate as an electron acceptor)	Differences in yield must be accounted for.
Deammonification (Anammox)	Growth and decay of Anammox bacteria	Appropriate inhibitions (i.e. nitrite toxicity) and limitations must be included.
pH	All significant equilibrium relationships (i.e. nitric and nitrous acid, ammonia and carbonate system)	pH modeling is essential because, for example, some inhibition effects are caused by unionized species concentrations.
Gas-liquid interactions	Stripping of certain model components such as ammonia and carbon dioxide	Gas-liquid interactions are essential to represent pH and in some cases, to properly represent growth-limiting conditions.

Sidestream Modeling Uses the Full ASDM Process Model

The reported models for the biological transformations described in the Introduction as important in sidestream processes were refined, and incorporated into the full General Activated Sludge/Anaerobic Digestion Model (ASDM). This model is described in detail in the "*Process Model Formulation*" Chapter. The model includes reactions for activated sludge, anaerobic digestion and sidestream reactor environments, as well as pH, gas transfer and chemical precipitation within the same model matrix. The model tracks over 50 components, with more than 80 processes acting on these components. The main processes are summarized below:

- Aerobic heterotrophic growth using complex substrate, acetate, propionate and methanol
- Anoxic heterotrophic growth on NO3- and NO2- using complex substrate and VFAs
- Anaerobic fermentation of complex substrate, propionate and methanol
- Growth of bio-P microorganisms and storage of polyphosphate
- Various hydrolysis, ammonification and colloid flocculation reactions
- Assimilative nitrate and nitrite reduction
- Anoxic growth of methylotrophs on nitrate and nitrite
- Growth of ammonia and nitrite oxidizer biomasses
- Growth of Anammox microorganisms
- Growth of autotrophic and heterotrophic methanogens
- Decay of all nine (9) active biomasses in different environments
- pH estimation based on the phosphate, carbonate, ammonia, acetate and propionate systems, including strong acids and bases, plus other relevant reactions
- Precipitation of various calcium, magnesium, aluminium and iron complexes (struvite, HDP, HAP, etc)
- Gas transfer of O2, CO2, N2, NH3, H2 and CH4 gases
- Inorganic suspended solids fixation during polyphosphate storage and heterotrophic growth

This complex model has been applied extensively for modeling the large number of processes occurring in suspended growth biological systems, and hence calibration requirements are significantly reduced.

The key difference between a sidestream reactor element and a bioreactor element, is that the sidestream reactor element has the **Local temperature** option selected by default as it is frequently used to model digester effluents that may be elevated in temperature. The default setting is a constant temperature of 35 degrees celcius. Also, the sidestream reactor element is seeded appropriately to improve solution speed.

Model Calibration

Calibration to High F/M nitrification rate tests

High F/M tests conducted to measure nitrification rates (WERF, 2003), often result in a significant accumulation of nitrite in addition to nitrate. Therefore, these tests provided excellent data for calibrating parameters related to nitritation (*i.e.* conversion of ammonia to NO_2) as mediated by ammonia oxidizing autotrophic bacteria (AOBs) and nitratation (i.e. conversion of nitrite to nitrate –NO_3) as mediated by nitrite oxidizing autotrophic bacteria (NOBs). These tests represented a case where ammonia concentrations were relatively high. Figure 1 shows a configuration that was used to simulate a High F/M test. The results from the simulation are compared to measured data in Figure 2

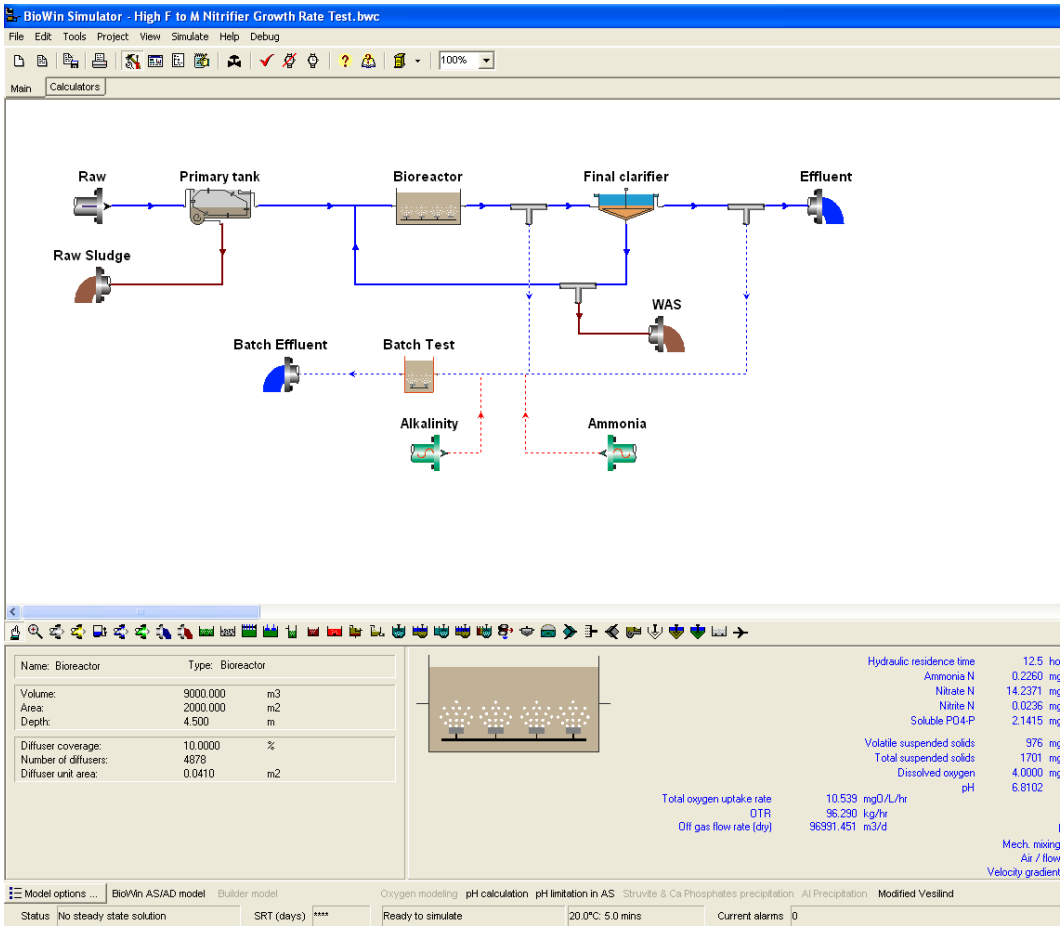


Figure 1: *BioWin configuration used to simulate a High F/M test for measurement of nitrification rates.*

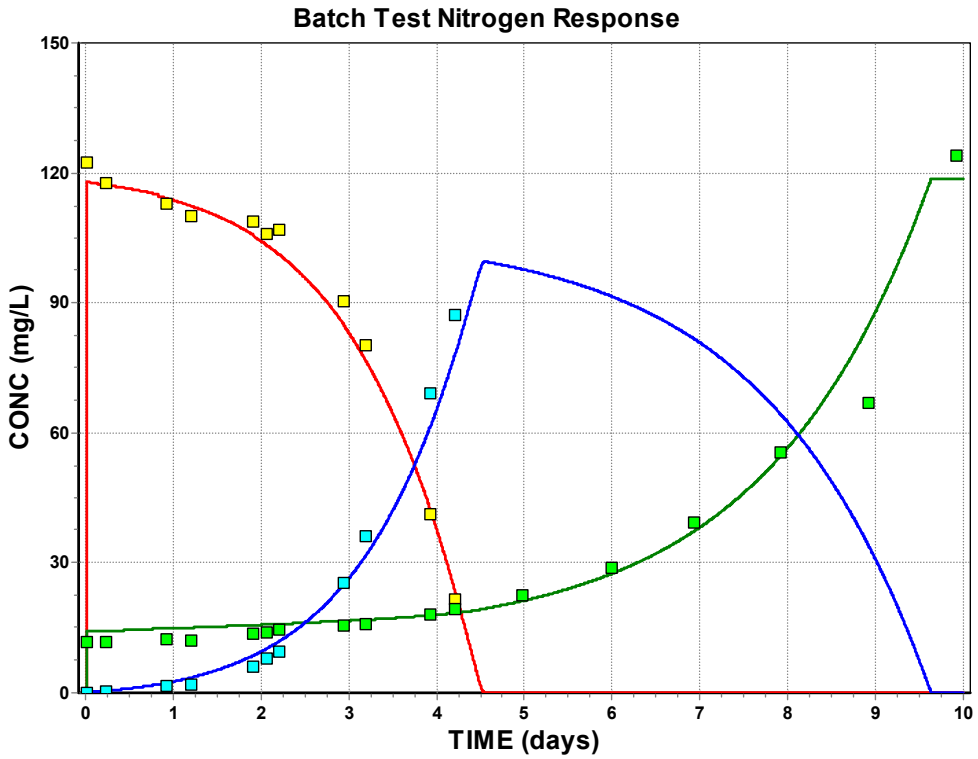


Figure 2: *BioWin simulation results and measurements from a High F/M for measurement of nitrification rates.*

Calibration to Low F/M nitrification rate tests

Low F/M tests conducted to measure nitrification rates (WERF, 2003), sometimes result in some accumulation of nitrite. In such cases, these tests provided excellent data for calibrating parameters related to nitritation (i.e. conversion of ammonia to NO2) as mediated by ammonia oxidizing autotrophic bacteria (AOBs) and nitratation (i.e. conversion of nitrite to nitrate –NO3) as mediated by nitrite oxidizing autotrophic bacteria (NOBs). These tests represented a case where ammonia concentrations were low.

The BioWin configuration used to simulate a low F/M nitrification rate test in a sequencing batch reactor is shown in Figure 3. The results from the simulation are compared to measured data in Figure 4.

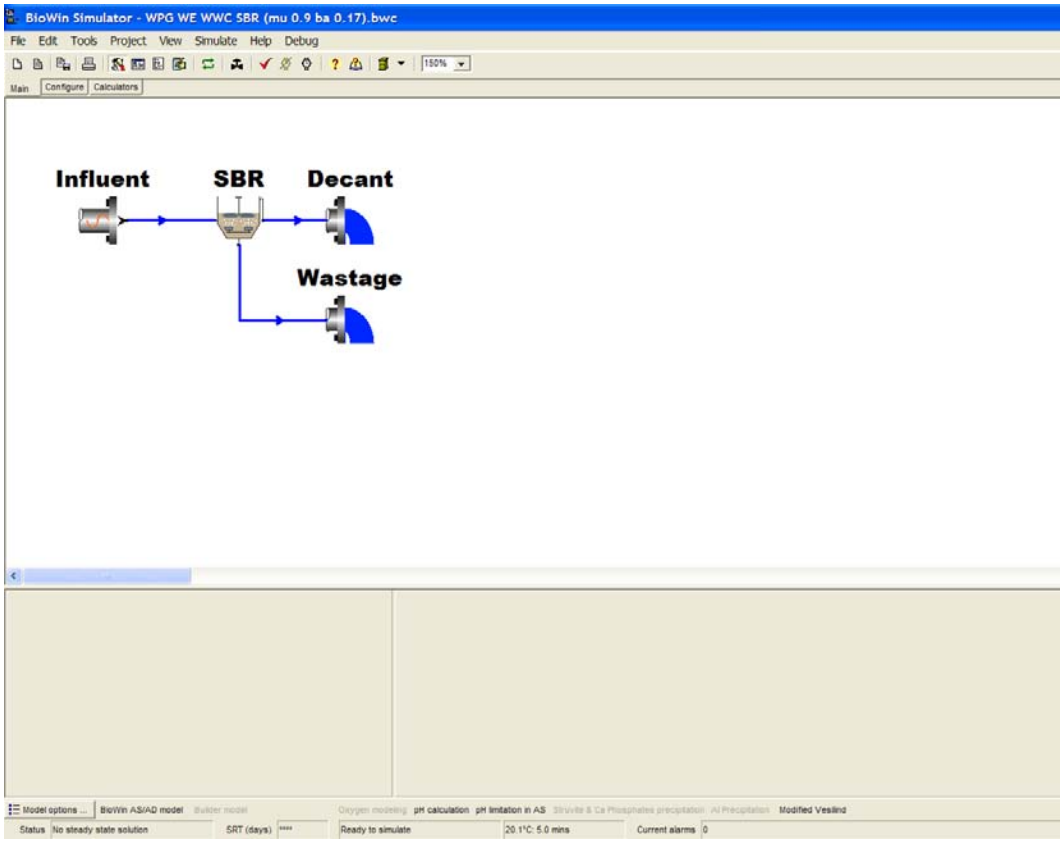


Figure 3: *BioWin configuration used to simulate a Low F/M test for measurement of nitrification rates*

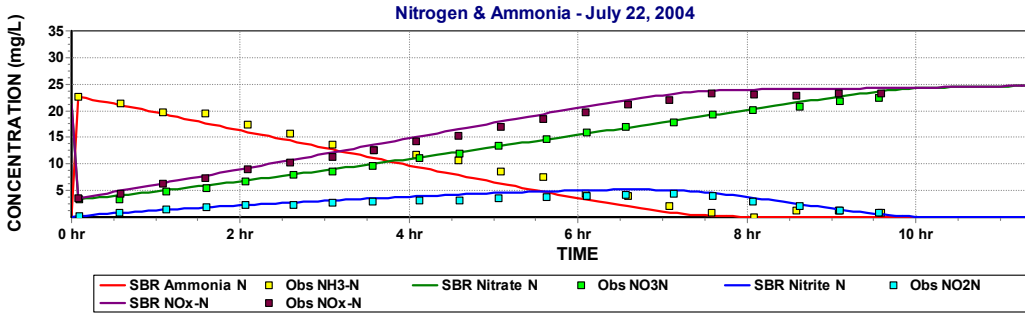


Figure 4: *BioWin simulation results and measurements from a Low F/M test for measurement of nitrification rates.*

Calibration to Sharon Process Data

Data published on a pilot-scale SHARON process (Caffaz *et al*., 2005) have been simulated to verify the ability of the model to predict nitritation only and the washout of NOBs under suitable conditions. The configuration used to simulate this process is shown in Figure 5. Table 2 lists some parameters on the system. Table 3 summarizes the characteristics of the centrate. The results from the simulation are compared to measured data in Figure 6.

Table 2: *Key design and operating information on a simulated nitritation reactor*

Reactor	Parameter	Value
Nitritation	Volume	7.4 m^3
	Dissolved Oxygen	3 mg/L
	Temperature	35°C

Table 3: *Characteristics of the centrate before treatment in the nitritation reactor*

Parameter	Value	Units
Flow	5.03	m^3/d
COD	286	mg/L
TKN	745	mg/L
Ammonia-N	730	mg/L
Total P	65	mg/L
TSS	67	mg/L

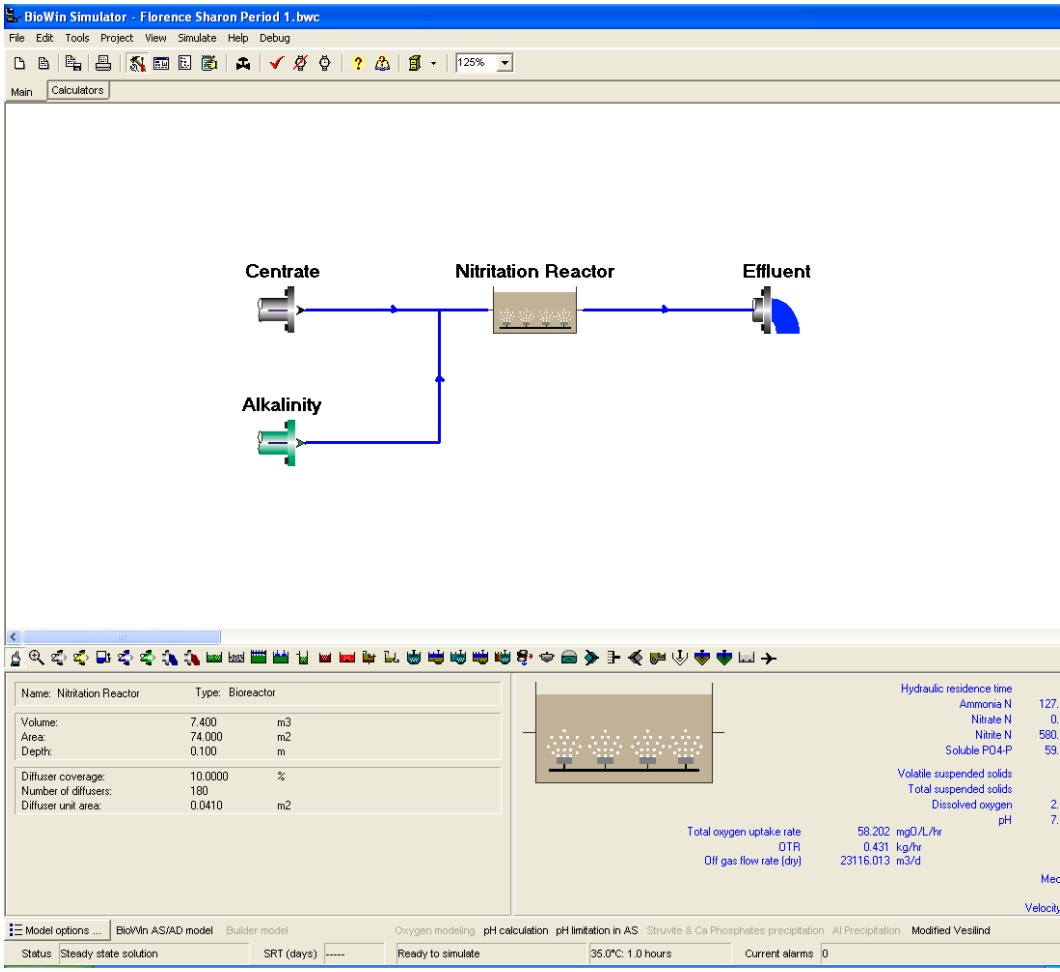


Figure 5: *BioWin configuration used to simulate a pilot scale SHARON process.*

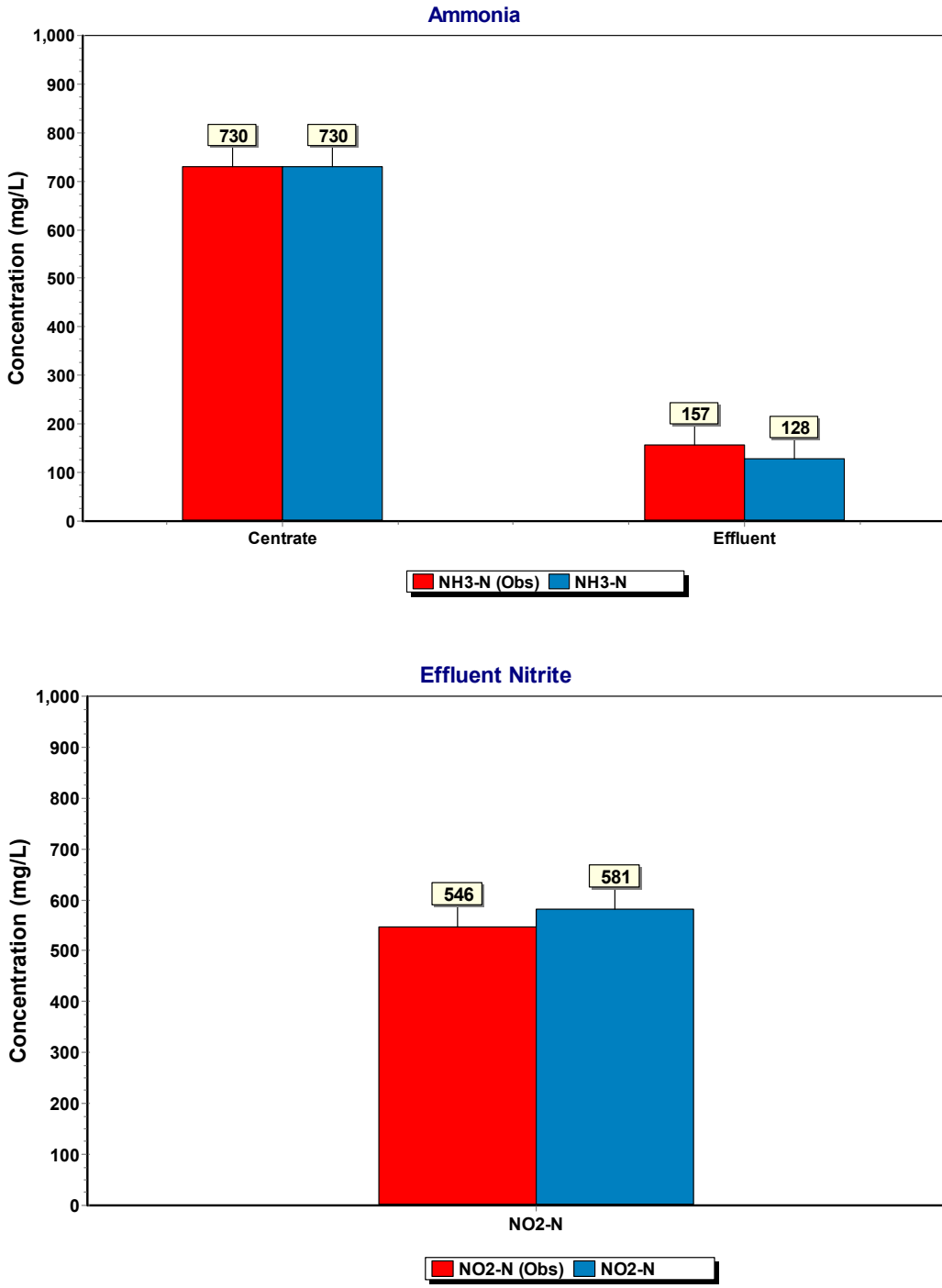


Figure 6: *BioWin simulation results and measurements for a pilot scale SHARON process.*

Examples

Nitritation-Deammonification System

A number of examples are available in the Pre-Configured File Cabinet. Figure 7 shows an example BioWin configuration used to simulate a nitritation-deammonification process. Key design and operating parameters for the sidestream treatment system are shown in Table 4. The operating strategy in this system is to achieve approximately 100% ammonia oxidation to nitrite in the nitritation reactor, rather than attempting to maintain partial ammonia conversion. The appropriate proportions of ammonia and nitrite in the feed to the deammonification reactor are achieved by adjusting the amount of centrate that bypasses the first reactor. In this arrangement, 40% of the centrate bypasses the nitritation reactor and flows directly to the deammonification reactor. Characteristics of the centrate flowing to the sidestream treatment process are shown in Table 5. Figure 8 shows the simulated nitrogen removal performance in this system.

Table 4: *Key design and operating information on a simulated sidestream treatment system*

Reactor	Parameter	Value
Nitritation	Volume	80 m^3
	Dissolved Oxygen	1 mg/L
	Temperature	31 ^oC
	Fraction of centrate bypassing nitritation reactor	0.40
Deammonification (Anammox)	Volume	350 m^3
	Dissolved Oxygen	0 mg/L (unaerated)
	Temperature	31 ^oC

Table 5: *Characteristics of the centrate before treatment in the sidestream process*

Parameter	Value	Units
Flow	144	m^3/d
COD	464	mg/L
TKN	427	mg/L
Ammonia-N	410	mg/L
Total P	493	mg/L
TSS	217	mg/L

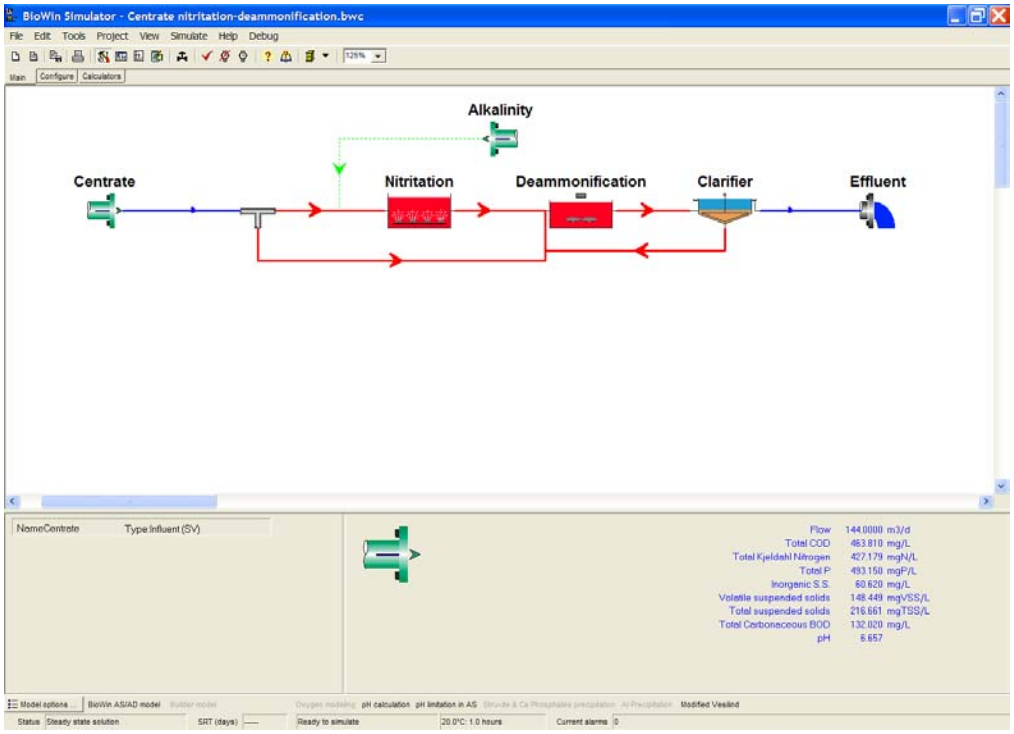


Figure 7**:** *BioWin configuration used to simulate a sidestream treatment system within a whole-plant configuration.*

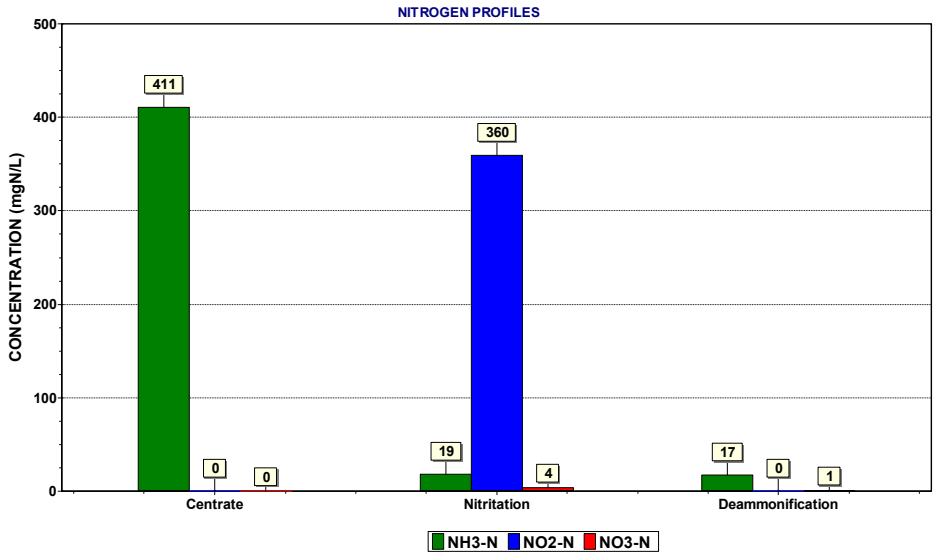


Figure 8: *BioWin simulation results for a sidestream treatment system within a whole-plant configuration.*

References

Caffaz, S., C. Lubello, R. Canziani, and D. Santianni (2005). "Autotrophic nitrogen removal from supernatant of Florence's WWTP digesters". Proceedings of the IWA

Specialized Conference Nutrient Management in Wastewater Treatment Processes and Recycle Streams, Krakow, Poland, 19-21 September, 2005.

Van Hulle, Stijn (2005). Modelling, simulation, and optimization of autotrophic nitrogen removal processes. Ph.D. thesis, Ghent University, Belgium.

Volcke, E.I.P. (2006). Modelling, analysis and control of partial nitritation in a SHARON reactor. Ph.D. thesis, Ghent University, Belgium.

WERF (Water Environment Research Foundation) (2003) *Methods for Wastewater Characterization in Activated Sludge Modeling*. Project 99-WWF-3, ISBN 1-893664-71-6. Alexandria, Virginia.

Wett, B. and W. Rauch (2003). The role of inorganic carbon limitation in biological nitrogen removal of extremely ammonia concentrated wastewater. Water Research, 37, pp. 1100-1110.