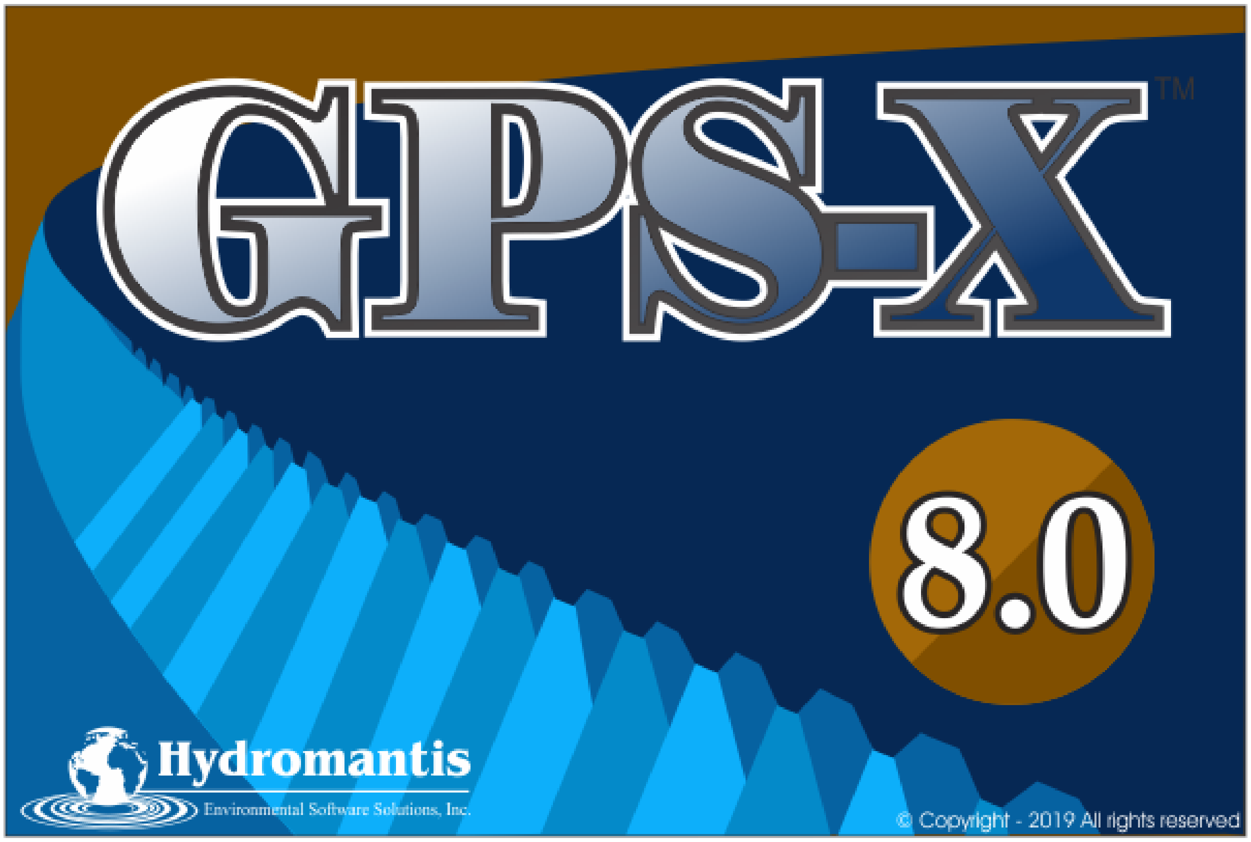
GPS-X User's Guide



GPS-X Version 8.0

Trademarks

GPS-X and all other Hydromantis trademarks and logos mentioned and/or displayed are trademarks or registered trademarks of Hydromantis Environmental Software Solutions, Inc. in Canada and in other countries.

ACSL is a registered trademark of AEgis Research Corporation

Adobe and Acrobat are trademarks of Adobe Systems Incorporated

Python is a registered trademark of the Python Software Foundation.

Microsoft, Windows, Word and Excel are trademarks of Microsoft Corporation.

GPS-X uses selected Free and Open Source licensed components. Please see the readme.txt file in the installation directory for details.

Hydromantis
Environmental Software Solutions, Inc.

Table of Contents

CHAPTER 1

GPS-X Preview

STARTING GPS-X

Start GPS-X by either double-clicking on the GPS-X v8.0 icon on the desktop, or by accessing the ***"Hydromantis GPS-X 8.0"*** program group in the Windows "Start Menu", and selecting "***GPS-X 8.0"***

ELEMENTS OF THE MAIN WINDOW

The basic elements of the GPS-X window are:

- Title bar
- Menu bar
- Toolbar
- Drawing Board
- Simulation Toolbar (in Simulation Mode only)
- Modelling/Simulation toggle button

A short description of each of these elements is provided below.

TITLE BAR

In addition to the buttons that control the appearance of the main window; the title bar area contains information about the version of GPS-X, the current layout being edited, and the library that this layout is using.

GPS-X 8.0 [<layoutname>] - <library>

MENU BAR

Most of the features of GPS-X are accessed from one of these six menus.

- File
- Edit

- View
- Layout
- Tools
- Library
- Help

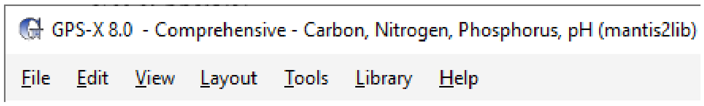


Figure 1-1 –Menu Bar

A brief description of each menu and their associated item(s) is provided below.

FILE MENU

The **File** menu contains items for performing file handling and manipulation.

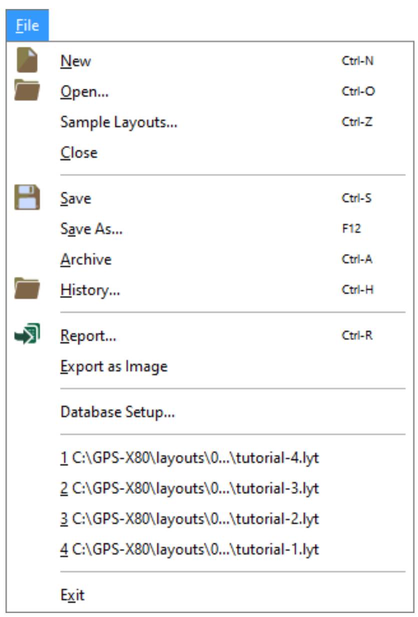


Figure 1-2 - File Menu

New

Create a new session with a blank drawing board.

Open…

Opens a file browser where you can browse to and select an existing GPS-X layout file.

A preview pane is on the right which shows you the drawing board image of the selected layout file.

The **Files of Type** drop down box can be used to show only GPS-X layout files (**.lyt** extension) or only GPS-X archived layouts (**.zip** extension).

Sample Layouts…

Allows you to select from the 50+ pre-configured layouts that come with a licensed version of GPS-X.

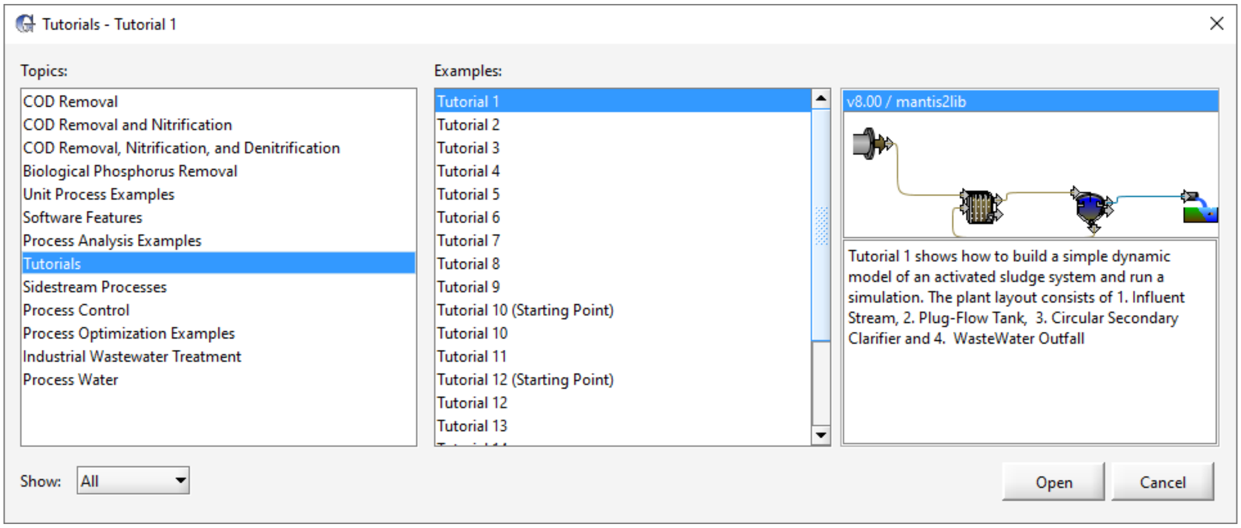


Figure 1-3 – Sample Layouts Dialog

Close

Closes the current layout. If any changes have been made, you are prompted about saving or discarding the changes.

Save

Saves the current layout to a file. This command will overwrite the existing layout file having the same name (shown in the title bar) with the current layout information.

Save As…

This opens a file browser, which allows you to enter a layout file name and location before saving. The current layout information is saved to the specified file.

Archive

Creates an archive file (i.e. zip compatible file) of all the GPS-X files pertaining to the layout currently open. This is useful when transferring files between computers, or when sending files as attachments via email.

History…

A tool for managing multiple versions of a GPS-X layout. The option **"Enable layout history"** must be checked in the ***View*** > ***Preferences*** > ***Layout*** menu to allow access to the layout history management tools. Further details on the **History** menu can be found in CHAPTER 2 (**Layout History Database**)

Report…

Create a report with information about the current layout. See the **Generating a Report** section in CHAPTER 7 for more information.

Print Layout

Used to send an image of the layout to a printer.

Database Setup

See the **Using an SQL Database with GPS-X** section in CHAPTER 13 for details.

Recent File List…

A list of recently-opened layouts is shown at the bottom of the **File** menu. The number of layouts shown here can be set in the ***View*** > ***Preferences*** >***Layout*** tab, under the **Settings** header.

Exit

Exits the program. You will be prompted to save or discard any unsaved changes to the layout.

EDIT MENU

The **Edit** menu includes items related to manipulation of objects on the drawing board.

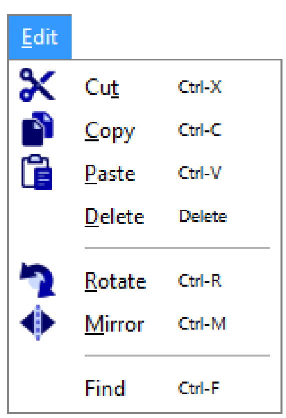


Figure 1-4 - Edit Menu

Cut

Cuts the currently selected process(es) from the drawing board and places it on the clipboard.

Copy

Copies the currently selected process(es) from the drawing board and places it on the clipboard.

Paste

Pastes process(es) from the clipboard onto the drawing board.

Delete

Deletes the currently selected process(es) from the drawing board.

Rotate

Rotates the currently selected process(es) 90 degrees in the counterclockwise direction.

Mirror

Horizontally flips the currently selected process(es).

Find

The **Find** menu item brings up the “Find” dialog, which can be used to search for GPS-X variables by entering part of the plain text or cryptic variable name.

VIEW MENU

The **View** menu button provides control over the look of the drawing board and how the object icons are displayed.

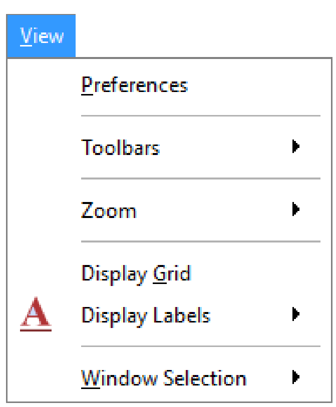


Figure 1-5 – View Menu

Preferences

The **Preferences** menu item brings up the “Preferences” dialog, which contains a series of tabbed sheets including the Layout, Input/Output, Build, and Python tabs. This window is used to set the default preferences for new layouts including the default library at start up:

Layout	details on default libraries, directories, etc., at startup, as well as settings for the look and feel of objects on the GPS-X drawing board.
Input/Output	default input controller and output graph types, plus details on report generation options.
Build	details on FORTRAN compiler and ACSL model build options.
Python	Details on the Python instance being used by GPS-X

Toolbars → Process Table

Gives you the ability to hide/show the process table.

The process table consists of a collection of unit processes and control points that are used to build the model layout for a wastewater treatment plant. One or more copies of each object in this table can be dragged onto the drawing board.

The objects are arranged in groups of similar unit processes, such as **Primary Treatment, Suspended Growth Processes**, and **Biosolids Treatment.** Click on the group name to open the group and display the available objects.

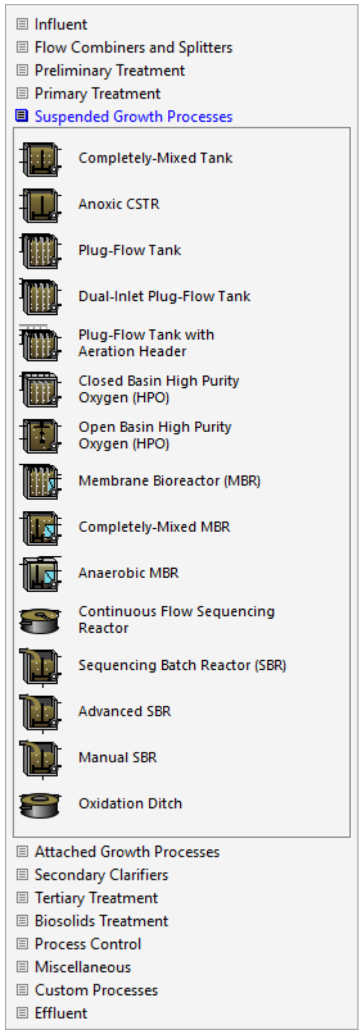


Figure 1-6 – Process Table

Zoom

The **Zoom** menu item contains a series of options for adjusting the level of zoom applied to the drawing board. The total drawing board consists of a grid 32 blocks wide by 32 blocks high, with each block capable of containing one object icon. The displayed area is normally only a small portion of the total area available, as most models require fewer than 20 layout objects.

The **Locator** option will open a window which provides a convenient way of zooming in or out of the drawing board. To zoom in on a specific region of the drawing board, click and drag the pointer within the **Locator** window around the area of interest. If the selected area is larger than the area displayed on the drawing board, the effect will be to zoom out.

The **Zoom to Selection/Plant** option is used to automatically zoom on the plant, such that an empty block appears around the edge of the plant. If an area has been selected on the drawing board, this option will zoom in such that the selected area fills the drawing board.

The **Zoom Out** option is used to zoom the drawing board out by adding a row of blocks to each edge of the drawing board.

The **Zoom In** option is used to zoom in on the plant by removing a row of blocks from each edge of the drawing board.

Display Grid

Gives you the ability to hide/show the grid lines on the drawing board.

Display Labels

Hides/shows the object and/or stream labels.

Each process's connection stream on a layout is automatically assigned a label.

In contrast, the processes themselves do not automatically receive a label, but may be assigned a label by the user. It is sometimes important to view the labels assigned to an object or connection point.

Window Selection

Used to select, bring forward, and give focus to any window in the modelling or simulation environment. It is useful for retrieving windows that may have fallen behind other windows during the setup of graphs and controllers.

LAYOUT MENU

The **Layout** menu includes items used to set global simulation parameters, user-defined customization code, and common plant wide properties.

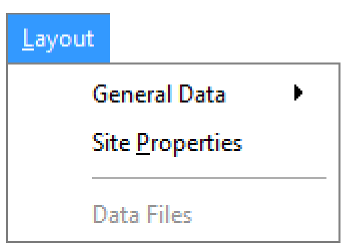


Figure 1-7 – Layout Menu

General Data

The **General Data** menu item contains a series of sub-menus including the **System, User** and **User Files** sub-menus.

The **System** sub-menu is used to gain access to the global simulation parameters, such as parameters related to the operation of the steady-state solver and the optimizer.

The **User** sub-menu is used to gain access to the user-defined variables.

The **User Files** sub-menu is used to define custom code and user-defined variables.

Site Properties

The **Site Properties** item allows users to customize the physical input parameters of the plant (Plant Site Properties tab) and the simulation date (Simulation Setup tab). Additional plant information can also be saved under the Plant Information tab.

Data Files

The **Data Files** menu is only available in **Simulation Mode** and is used to manage the files used in the active scenario (see **Using Scenarios**). From this menu you create new datafiles, add existing data files to the scenario or remove data files from the scenario. You can also edit data files, but if the file is used in multiple scenarios, it will change the information in all of them.

TOOLS MENU

The **Tools** menu includes items related to the setup and use of a plant model.

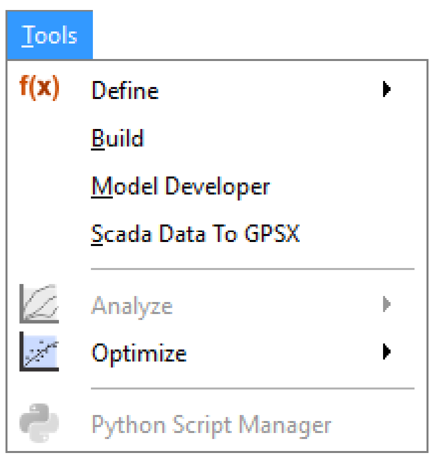


Figure 1-8 – Tools Menu

Define

Allows you to interactively specify an equation for calculating one of six different layout-wide variables including the **Solids Retention Time (SRT)** and **Food/Microorganism ratio (F/M)**. In practice, it is found that formulation of the defining equations for these variables is plant specific. The **Define** feature allows you to interactively specify how these variables are to be calculated for a specific layout.

Build

There is an "auto-build" feature which knows when to build and rebuild model code. However, if users wish to force a model build, it can be done from this menu.

This translates the flow sheet to binary executable code. GPS-X uses a special procedure to convert the graphical images in the drawing board first to a high-level simulation language (ACSL) code and then to a FORTRAN binary executable program.

Analyze

(This feature is available only to those who have purchased the Analyzer module)

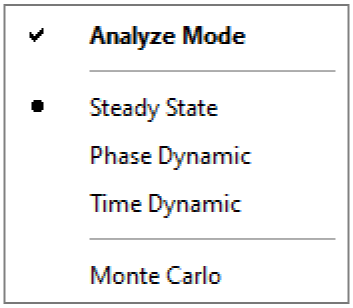


Figure 1-9 – Analyze Menu

Used to test the 'model validity' by conducting sensitivity analyses on selected parameters.

The Analyzer is turned 'on' by selecting the Analyze Mode checkbox in the **Analyze** sub menu, or by clicking on the **Analyze** button on the toolbar.

With the **Analyzer** active, four options exist for the analysis:

Steady State	conducts a steady state simulation for different values of independent variable
Phase Dynamic	calculates state variables at a specified point in time for different values of the independent variable.
Time Dynamic	conducts a dynamic simulation for each value of an independent variable for a specified time interval.

Monte Carlo conducts a Monte Carlo analysis of an independent variable for a specified range and probability distribution.

Optimize

(This feature is available only to those who have purchased the Optimizer module)

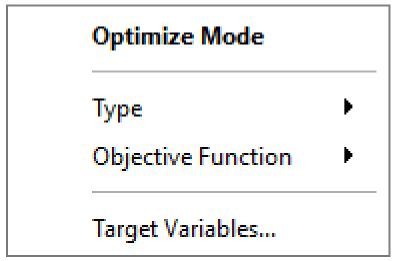


Figure 1-10 – Optimize Menu

A flexible, dynamic optimization package for evaluating important model parameters. Given appropriate dependent data, and a user-defined objective function, GPS-X can identify an optimal value for a selected model parameter.

The **Optimize** sub-menu controls the set up and operation of the optimization tools.

In Modelling mode, selecting the Optimize Mode will start a wizard that will set you through the process of setting up the optimization.

In Simulation mode, selecting the Optimize Mode will inform the program that the simulation that is about to be performed should use the optimizer feature.

Optimizer Setup Wizard

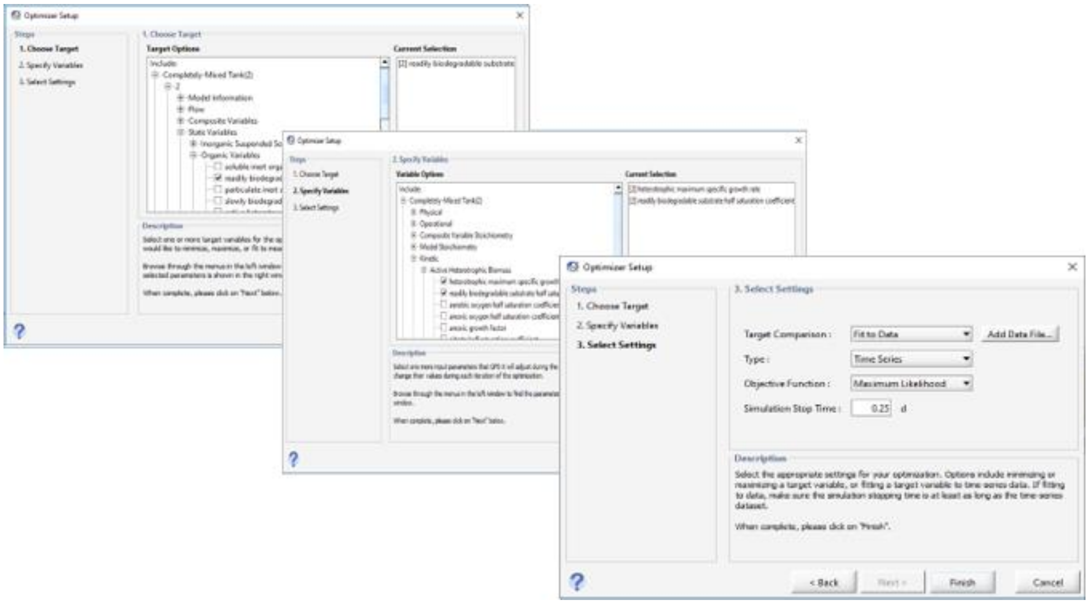


Figure 1-11 – Optimizer Setup Wizard

An optimizer setup wizard is used for specifying target variables, optimize variables, and optimizer settings. Once these are specified, you must switch to simulation mode where GPS-X automatically creates a input parameter menu with the optimized parameters and creates an output graph of the target variables.

Type

Two options are available from this sub-menu:

Time Series	optimizes based on a set of one or more time series of dependent data
DPE	**(Dynamic Parameter Estimation)** – optimizes a moving time window of data for use in on-line model calibration. (This feature is available to those who have purchased the ***Advanced Tools*** optional module.).

The data requirements and form of the objective function depend on the type of optimization being performed.

Objective Function

You have the flexibility of specifying the form of the objective function to use. This sub-menu contains several objective function options including:

Absolute Difference creates an absolute error form of the objective function.

Relative Difference creates a relative error form of the objective function.

Sum of Squares creates and absolute sum of squares error form of the objective function.

Relative Sum of Squares creates a relative sum of squares error form of the objective function.

Maximum Likelihood creates a maximum likelihood from of the objective function.

In the objective functions, the errors associated with a target variable are the differences between the predicted values and the observed values.

Target Variables

This menu item displays a list of the target variables selected for the optimization. You can use this item to verify the target variable names.

Python Script Manager

This menu item brings up the “Python Script Manager” Window. The **Python Script Manager** is only available when GPS-X is in Simulation mode. This window is used to edit or run existing Python scripts in GPS-X and create new Python scripts.

LIBRARY MENU

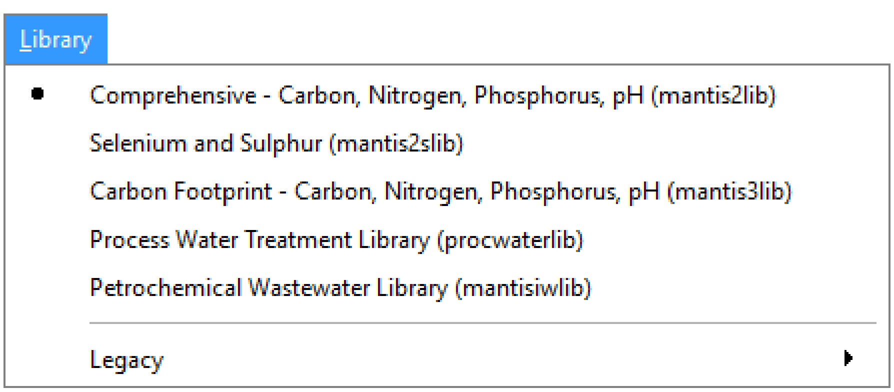


Figure 1-12 – Library Menu

The library menu is used to select/view which library the current layout used. View the **Technical Reference Manual** for more information about libraries.

HELP MENU

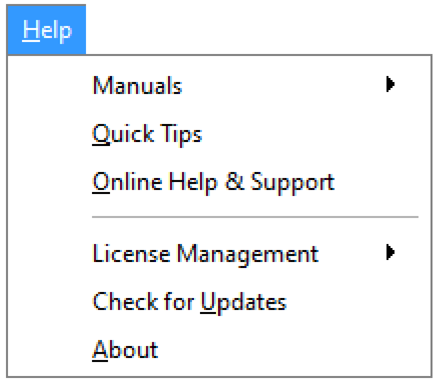


Figure 1-13 – Help Menu

Manuals

The **Manuals** menu item contains links to the GPS-X companion literature. Selecting the **User's Guide**, **Tutorial Guide**, **Model Developer Guide**, and **Technical Reference** will display the corresponding GPS-X manual in electronic format using Adobe Acrobat Reader.

Quick Tips

The **Quick Tips** menu item will open a GPS-X Quick Tip. These tips highlight GPS-X features you may be unaware to help you enhance your GPS-X user experience.

Online Help & Support

The **Online Help & Support** menu will link you to Online Help and Support page of the Hydromantis website.

License Management

The **License Management** menu item contains sub-options including **License Activation…**, **License Transfer…** and **View License**.

The **License Activation…** menu item is used to activate a product key that you've received from Hydromantis.

The **License Transfer…** menu item opens a window with instructions on how you can transfer your GPS-X license between computers.

The **View License…** menu item opens the local website where you can view any current licenses that are installed on your computer, their expiry date, and various other settings.

Check for Updates

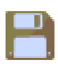
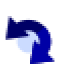
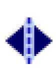
The **Check for Updates** menu item will verify that you are using the latest version of GPS-X

About

The **About** menu item will display information on this version of GPS-X.

MAIN TOOLBAR

The buttons are listed below. Not all of the buttons are available in both Simulation and Modelling Modes. For information about the actions that they perform, see the corresponding menu item description in the previous section.

	New Layout
	Open Layout
	Save Layout
	Cut
	Copy
	Paste
	Rotate
	Mirror
	Zoom to Selection/Plant
	Display Labels
	Define

	Analyze
	Optimize
Units: SI/US	Default Unit Set
	Data Files
	Report

DRAWING BOARD

The drawing board is the large area in the center of the main window. This is where you define your treatment plant process flow diagram. See **Creating Model Layouts** in CHAPTER 2 for a description of how to do that.

The Site Properties button can be found in the upper left-hand corner of the drawing board. See **Site Properties** in **CHAPTER 4** for a description of its use.

MODELLING/SIMULATION MODE

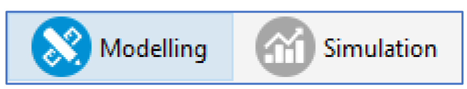


Figure 1-14 – Modelling/Simulation Mode Buttons

The **Modelling/Simulation Mode** button allows the user to switch between the two modes. It also activates the automatic rebuilding of layouts (if applicable).

Modelling Mode	For creating and/or editing models. Users must be in Modelling Mode to draw and change models on the drawing board, and define new variables or source objects. Any user-defined code (see Customizing GPS-X) must be entered while in this mode.
Simulation Mode	For using the model, and doing analyses. Users must be in Simulation Mode to run the model, create and use scenarios, create and use input and output graphs, and use the analyze/optimize functions. The GPS-X layout drawing

board will appear with a grey background when in Simulation Mode.

SIMULATION TOOLBAR

The **Simulation Toolbar** is only displayed in Simulation Mode. It is at the bottom of the main window.

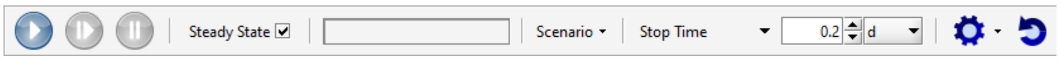


Figure 1-15 – Simulation Toolbar

Simulation Progress Controls

Start Runs the simulation from the beginning.

Resume If a simulation has been paused, this will resume the simulation from that point.
If a simulation has reached it's 'Stop Time', then you can increase that value and then continue the simulation.

Pause Pause a simulation.

Steady State

This checkbox controls whether or not to use the steady state solver.

Convergence/Simulation Progress Bar

This progress bar shows either the steady state convergence percentage (grey) or the simulation progress (blue).

Figure 1-16 – Convergence/Simulation Progress Bar

Scenario

This menu contains scenario-related commands including the commands to select, create, delete and show the contents of scenarios. See **Using Scenarios** section in CHAPTER 8 for more information.

Stop Time/Communication/Delay

This area of the simulation toolbar allows you to edit the stop time, communication interval, and the delay value. These three parameters can be changed interactively as the simulation proceeds.

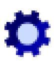
Click on the label to switch between the three variables.

Stop Time is the amount of time that the simulation will run before stopping. This value is also used to set the maximum time on time series graphs.

Communication interval is the period of time between communication with the model and the graphical interface. This interval affects both the spacing between data points on a graph and updating the model with the values of the input controllers. It can be any positive value greater than zero.

Delay is an artificial delay inserted in the simulation routine, which slows the simulation. For some models, this may be necessary to allow enough time to change controls and observe responses.

Simulation Control



This menu contains various settings regarding the simulation and some different low-level commands that can be sent to the model. For more information about these options, see the **Simulation Control** section in CHAPTER 8.

Reset

The **Reset** button can be used to clear all the previous simulation iterations and results. This feature keeps all the layout settings but resets the ***Initial Conditions***. Most people do not need to use this feature because simply starting the simulation over again (by pressing the **Start** button) will begin the simulation in an adequate place.

CHAPTER 2

Creating Model Layouts

LAYOUTS

GPS-X layouts contain data on the physical objects being modeled. Among other things, these data include the model(s) being used, physical dimensions, and connectivity between unit processes. The layout defines the structure of your plant model, including the kinds of plant behavior you can model, the degree of flexibility in setting up simulations, the model parameters you can change, and the response variables you can plot.

You can create as many layouts as you wish, with as many different configurations as you would like to model. Layouts and their associated data can be saved for later use, and the information in the layout can be exported for reporting purposes. This information and the layout schematic can be printed or copied to other documents so that you can prepare reports and papers. The layout contains all the information needed to reproduce a session.

Layouts usually correspond to and are prepared from the treatment plant process flow diagram. To prepare a layout of a treatment plant, you will need to know the configuration of the plant, and the unit processes that make up the plant. It is a good idea to refer to your plant diagram as you examine the procedures for building a GPS-X layout.

Opening an Existing Layout

To open a GPS-X layout file, complete the following steps:

1. **Click on the *Open* button.**

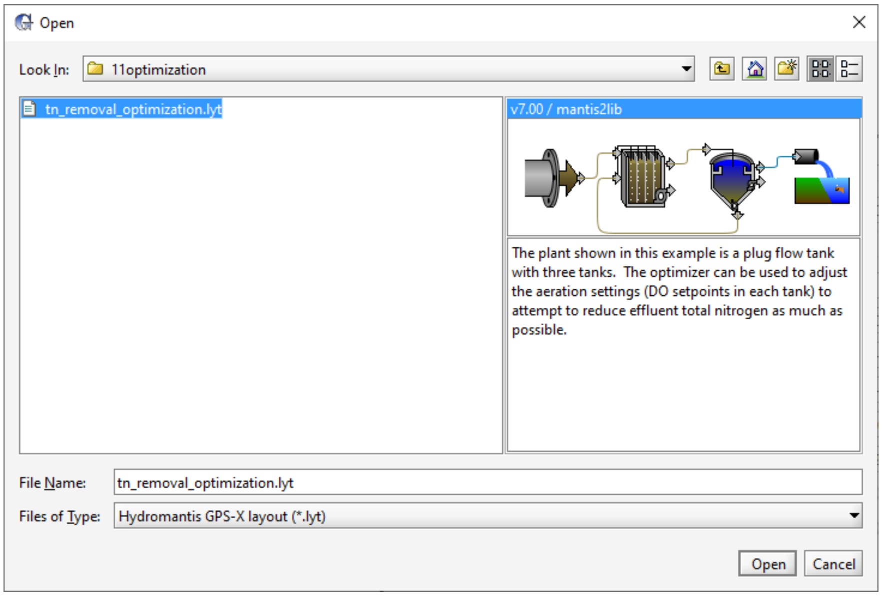


Figure 2-1 – Open File Browser

2. **Use the File Browser** to navigate and select the desired GPS-X layout file.
3. **Click the *Open* button**. The layout will be opened.

Before opening the model layout, GPS-X checks to see if the current layout has been saved. If the layout has not been saved, a warning dialogue box is displayed which provides the opportunity to save the layout.

If the layout that you are opening contains variables that are not defined in the applicable GPS-X library, a pop-up text form will be displayed that indicates the variables were mapped.

Creating a New Layout

A new layout is started by selecting **New** from the **File** menu, or by clicking on the **New** button on the tool bar.

Saving a Layout

Before getting started on the construction of a layout, it is best to save the layout with a new file name. As you make changes to the layout, you can select **Save** from the **File** menu or click on the **Save** button on the toolbar to update the layout file. If at any time, you want to save the layout to a different file name, select the **Save As…** item.

To create a new layout, complete the following steps:

1. **Open the *File* menu and select the *Save As…* item**.
2. **Use the file browser** to navigate to the directory where the file is to be saved.
3. **In the *File Name* text entry field,** enter a layout name. Layout names can consist of any alpha-numeric character (among other exclusions, spaces and periods are not allowed). You do not need to append a file extension as one is automatically added to the layout name you enter (**.lyt**)
4. **Click on the *Save* button**. The layout will be saved and the layout name will be displayed on the main window's title bar. In general, it is a good idea to save your work often to avoid any loss of information.

GPS-X saves all the information needed to restore your modelling session to the state it was in when you last saved the file. This includes the layout objects and their connectivity, any controls you may have defined and the graphical output you specified. There are several other data files, which are sometimes used with a layout, and in most cases, these are associated with the layout name rather than stored within the layout file. These additional files include raw data files, code additions and output data files. Details on these other files are provided in subsequent chapters.

SELECTING AND PLACING OBJECTS

You are now ready to begin building a model layout. GPS-X allows you to build models graphically by selecting object icons from the objects in the **Process Table** and placing them on the drawing board. The resulting graphical representation of the plant is later converted to a dynamic model. To be able to select from the objects, you must first display the process table.

To hide/show the process table:

1. **Select the *View* > *Toolbars* > *Process Table* menu item.** The **Process Table** window is displayed on the left-hand side of the drawing board. The objects are grouped into collections of like unit processes. Click on the group name (i.e. "*Attached Growth Processes*") to open the group.

A distinction is made between **object**, **icon**, and **model**. ***Object*** is used to refer in general to a unit process or control point (including the icon and model), ***icon*** is used to refer to the picture, and ***model*** refers to the underlying mathematical formulae. This distinction is important for understanding the way layout information is stored and how you can tailor GPS-X to include your own icons and models. A complete examination of these topics is given in CHAPTER 11.

Each object has one or more models. You manipulate icons on the drawing board to build a schematic representation of your plant. The icons in the layout are images, which serve as a visual reminder of the type of object being modeled rather than the model itself. Each icon is linked to one or more models, which are manipulated in the preparation of the full plant model.

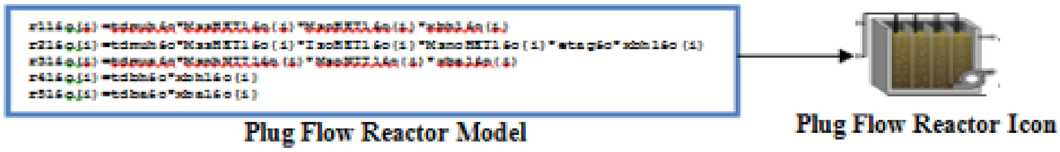


Figure 2-2 – A Distinction is made between an Object and its Attributes

Once the process table is displayed, you can begin to place objects on the drawing board.

To select an object and place its icon on the drawing board:

1. In the **Process Table**, left-click on any object group.
2. **Press and hold** the left mouse button on any object icon and drag the object to the drawing board.
3. **Release** the mouse button. A copy of the object's icon will be dropped onto the drawing board.

You can continue to drop object icons onto the drawing board by repeating this operation. Notice that the drawing board is divided into equally sized cells. The entire drawing board area is a 32x32 grid. Each of the cells can contain one process icon. Select the **Display Grid** button on the toolbar to see the gridlines.

Once an icon has been dropped onto the drawing board area, an alternative way to add a new object is to select the object in the drawing board area by clicking and dragging the mouse across the object of interest. This operation shades the selected grid cells a light blue color. Once an area of the drawing board is selected it can be cut, copied or deleted as required. If the objects were cut or copied, they can be pasted to a different area of the drawing board to create exact copies of the originals (including any attributes you may have changed in the originals). Changing object attributes is discussed in CHAPTER 4.

CONNECTING OBJECTS

When building a plant layout, you first decide which object(s) to model and then place the appropriate icons on the drawing board as described in the previous section. Having done so, it is necessary to establish the flow paths between process objects. This connectivity is important as it is needed by GPS-X to develop the material balances from which dynamic model equations are prepared.

Icon Connection Points

Each icon has a number of connection points. These points are easily recognized since they are drawn as horizontal or vertical arrows projecting from the icon.

Numbers are assigned automatically to connection points as their labels. Each stream label must be unique to the layout. You can easily change these labels as described in the **Labels** section of CHAPTER 4.

After the icon is placed on the drawing board, the connection points become *active*, that is, if you move the mouse pointer over a connection point, the arrow pointer disappears and a connecting arrow is drawn at the connection point.

As shown in **Figure 2-3**, when the mouse pointer is placed over a connection point, a small connecting arrow is displayed (right side of plug flow reactor, connection point 3).

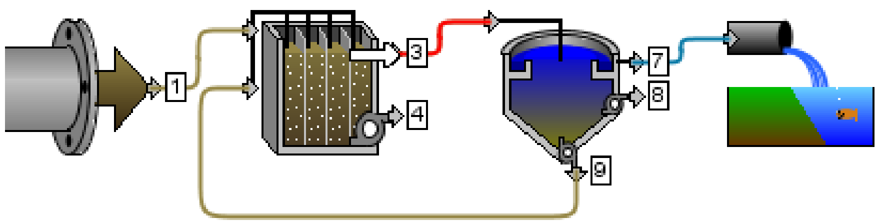


Figure 2-3 - Model Layout Showing Connection Points

(Reactor Effluent Connection Point is Selected)

Specifying a Flow Path

A flow path is specified by joining the connection points of two objects. To specify a flow path, complete the following steps:

1. **Move the mouse pointer to the icon connection point, which you would like to be the flow source**. The mouse pointer changes to a connection arrow when it is over an icon connection point.

2. **Press the left mouse button and drag a connecting pipe to the icon connection point which you would like to be the flow terminating point**. The mouse pointer changes to a connection arrow when it is over an icon connection point.

3. **Release the mouse button**. With this action, a flow path has been defined.

To delete a flow path, complete the following steps:

1. **Move the mouse pointer over the beginning or ending connection point of the flow path you would like to delete**. The mouse pointer changes to a connection arrow when it is over an icon connection point.

2. **Press the left mouse button and drag a connecting pipe to an empty grid cell.**

3. **Release the mouse button**. You will be prompted to confirm the deletion of the flow path.

To move a flow path, complete the following steps:

1. **Move the mouse pointer over the beginning or ending connection point of the flow path you would like to move.** The mouse pointer changes to a connection arrow when it is over an icon connection point.

2. **Press the left mouse button and drag a connecting pipe to the new destination of the flow path.**

3. **Release the mouse button**

Moving Flow Paths Around

Once a flow path has been correctly drawn on the drawing board, the location of the line connecting the two objects can be moved around to make the diagram clearer and/or less cluttered.

To move an existing flow path while maintaining the connections to the objects, complete the following steps:

1. **Place the mouse pointer over the flow path line** somewhere between the beginning and ending connection points. The pointer will change to an up/down or left/right arrow, depending on the orientation of the line.

2. **Hold the left mouse button while dragging the line to another location.**

3. **Release the mouse button**. This action can be done as many times as required to create as many corners as needed.

Normally, you will want to place the desired object icons on the drawing board and then specify the flow paths. GPS-X contains logic to guide you in selection of appropriate flow paths. For example, GPS-X does not allow flow paths which initiate at an icon input and terminate at an icon output. Similarly, you cannot have flow paths initiating and terminating on the same icon. This logic prevents the creation of incorrect layout diagrams and ensures that a valid model can be prepared. It is not necessary to specify flow paths to every connection point on every icon. For example, in some layouts, one or more objects have zero influent flow, and therefore, do not have any flow path connected to the influent connection point.

Flow Path Color

The color of the flow path is used to quickly convey information about the stream. There are three possible colors that a stream can be: Blue, Brown and Black.

Streams that are colored Blue represent flows of treated water. This includes any streams treated by a tertiary treatment object, streams coming from a clarifier overflow or a water influent.

Streams that are colored Brown are used to represent the flows of wastewater. This includes any clarifier underflow lines, suspended growth effluents and wastewater influents.

Streams that are colored Black are used to represent any streams where the treatment quality of the water cannot be immediately be determined from looking at the object. This includes flow combiners, flow splitters and black box objects.

EDITING THE LAYOUT

At some point in building a plant layout it may become necessary to add or remove object icons and flow paths, either to investigate alternative layouts or correct an existing one.

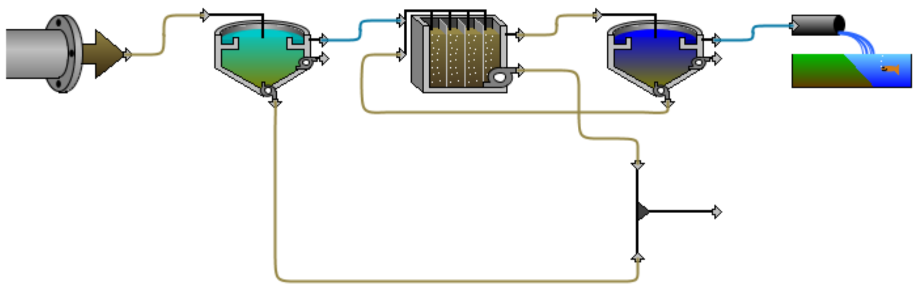


Figure 2-4 –Layout of a Conventional Activated Sludge Plant

Figure 2-4 shows the completed layout for a conventional aerobic suspended (activated sludge) treatment process. The layout consists of a single influent, a primary settler, a plug-flow reactor, a secondary clarifier and an effluent outfall structure. Wastage flows from the primary settler and the plug flow reactor are connected in the layout to a combiner object, which consolidates the flows.

Consider the types of editing operations you might want to perform given a layout such as the one in **Figure 2-4**. All or part of the layout may be similar to another plant process train that you would like to model, so it would be convenient to ***copy*** this portion. If you want to compare layouts with different types of reactor models, you might want to replace the plug-flow reactor with a complete mix reactor. Not all plants waste from the aeration tank, so you may want to ***delete*** this flow path. Some plants don't have a primary settler, or you might want to compare plant performance without one, in which case you would want to ***delete*** this unit process icon.

The placement of icons may not be exactly as you would like so you may want to ***move*** one or more of the icons. In some cases, it might be necessary to move a flow path, that is, change the initial and/or terminal points. For example, you might want to move the wastage flow path so that wastage occurs from the secondary clarifier rather than the plug flow reactor.

In some cases, editing changes affect the model structure; whereas, in other cases, the changes are only cosmetic. For example, replacing a plug flow reactor with a completely mixed reactor significantly changes the plant model, but moving the same reactor to another location on the drawing board (assuming the flow paths are not also changed) does not affect the formulation of the model.

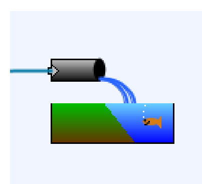
A cosmetic change, which is often useful, is ***rotation***. GPS-X provides facilities for changing the orientation of object icons in order to modify the placement of flow streams. GPS-X uses a special mapping technique to draw the flow paths between object connection points. Rotating an icon can result in improved placement of the flow paths for complex drawings containing many flow paths.

Some editing operations can be done in blocks. This is convenient when making large changes to the layout. GPS-X allows block copy, cut and paste, and block delete. If you have several parallel trains that are identical, you can use the block option to clone a single train and create the remaining trains. Large parts of a layout can be removed by dragging out a region in the block-delete, and you can easily move a large block to a different location using block-cut and paste. These operations are described in detail below.

Selecting One or More Drawing Board Objects

To select a single object:

1. **Select the object in the drawing board area by clicking the mouse on the object of interest**. This operation shades the block on the drawing board containing the object light blue.



To select a block of objects:

1. **Select an area of the drawing board by clicking and dragging the mouse across the area of interest**. This operation will shade the selected blocks light blue.

Once an area of the drawing board is selected, the objects within that area can be deleted, copied, or cut and pasted.

Deleting

To delete objects from the selected area of the drawing board:

1. **Select an area of the drawing board by clicking and dragging the mouse across the area of interest.** This operation shades the blocks containing the selected object(s) light blue.

2. **Select Edit > Delete from the main menu** or hit the Delete key on your keyboard. A warning message will be displayed for you to verify the delete operation. Selecting ***Yes*** causes the object icon and its underlying model to be deleted. If you select ***No***, the object will not be deleted.

Copying and Cutting

To cut or copy a selected area of the drawing board:

1. **Select an area of the drawing board by clicking and dragging the mouse across the area of interest**. This operation will shade the selected blocks light blue.

2. **Click the applicable *Cut/Copy* button on the toolbar**. This operation stores the object data on the local clipboard.

Pasting

To paste an object(s) stored on the clipboard:

1. **Click on the cell of the drawing board where the object(s) is to be pasted**.

2. **Click the *Paste* button on the toolbar**. The cut or copied object(s) will be pasted to the selected cell.

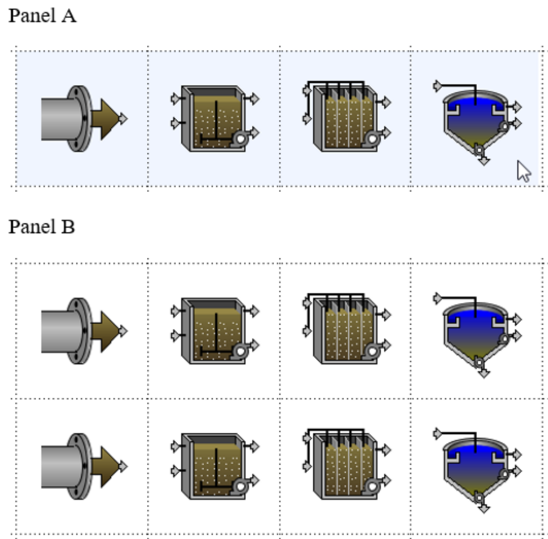
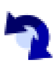


Figure 2-5 – Performing a Block Copy Operation

Rotating



Sometimes it may be advantageous to change the orientation of the icon. In some cases, this may be visually necessary to adjust the placement of flow paths. This is often the case with flow combiners and splitters. Connections between objects are maintained when the icon is rotated.

To rotate an icon:

1. **Select the object that is to be rotated.**
2. **Click the *Rotate* button on the toolbar**. The icon's orientation will change 90 degrees. Continue clicking the **Rotate** button until the desired orientation is achieved.

Mirroring

Yet another cosmetic change that is sometimes needed is mirroring an object icon.

To mirror an icon:

1. **Select the object that is to be reversed.**
2. **Click the *Mirror* button on the toolbar**. The object's icon will be flipped horizontally.

Zooming and Panning

After placing several objects on the drawing board, you may want to adjust the level of zoom to reduce the amount of whitespace on the drawing board or to focus on specific objects in the plant. This can be done using **Zoom to Selection/Plant**.

To zoom in on the plant:

1. **Click on the Zoom to Selection/Plant icon**. GPS-X will adjust the level of zoom on the drawing board so there is an empty block framing the plant on each side.

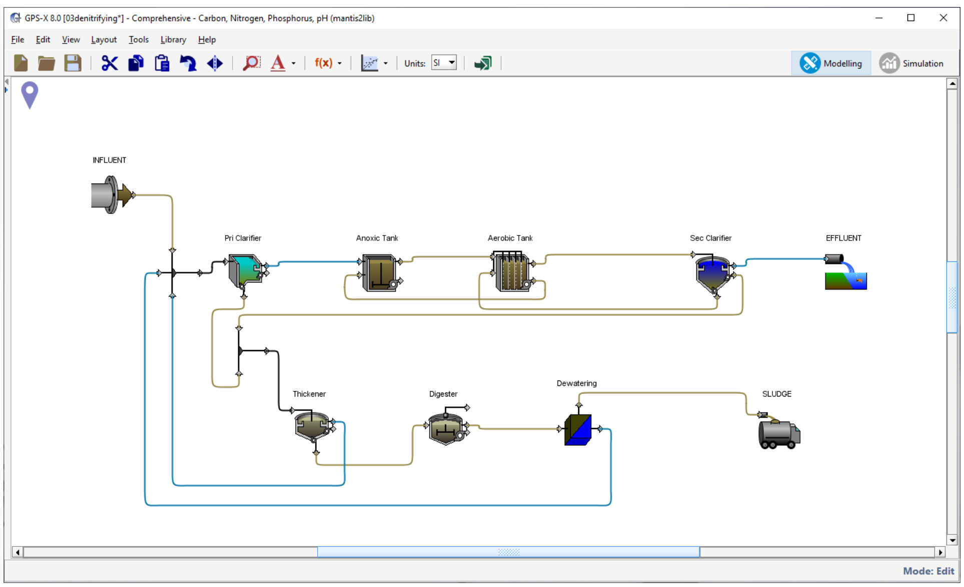


Figure 2-6 - Zoom to Plant Scaling

To zoom in on objects of interest:

1. Select an area of the drawing board by clicking and dragging the mouse across the area of interest. This operation will shade the selected area light blue.

2. **Click on the Zoom to Selection/Plant icon**. This will zoom in on the selected area so that it fills the entire drawing board

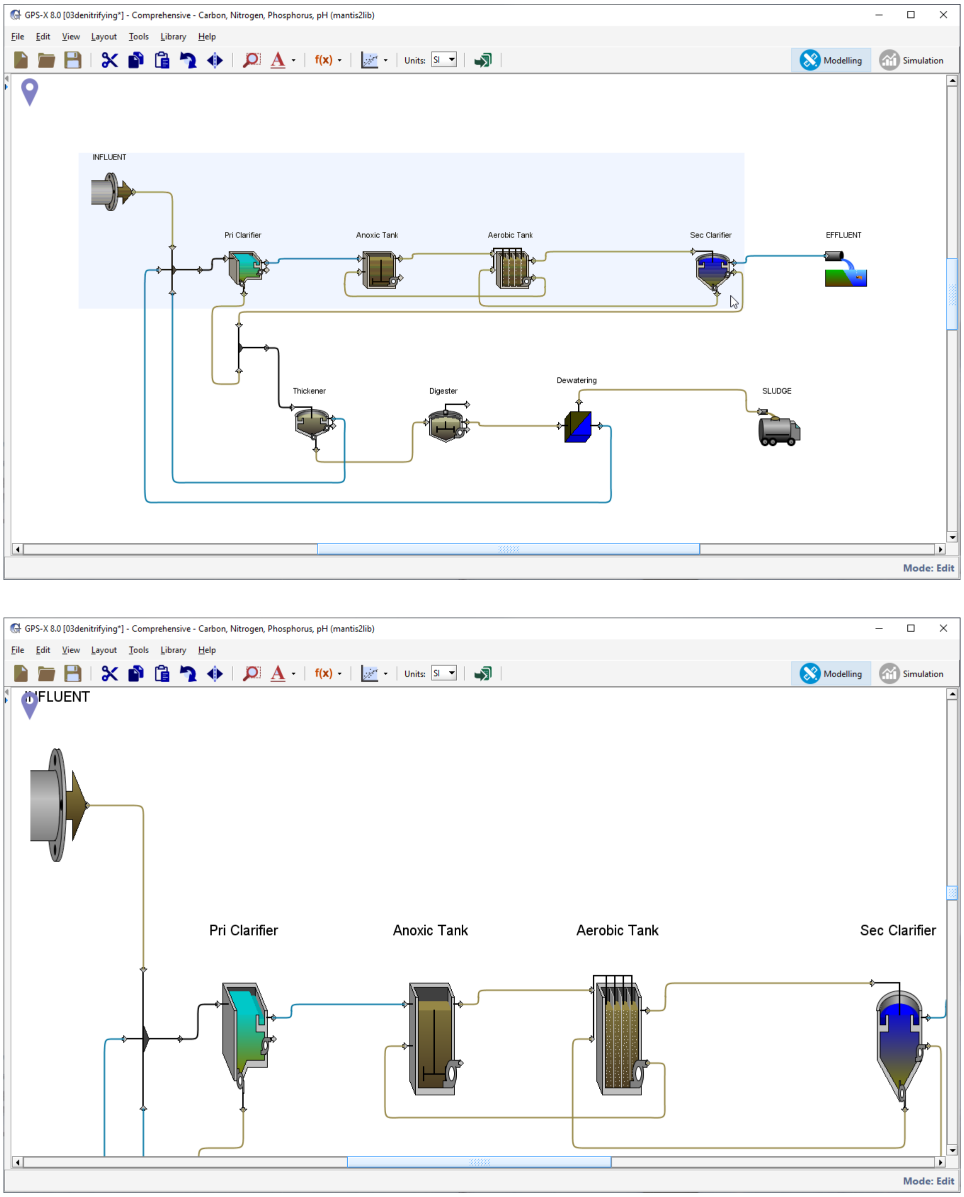


Figure 2-7 - Zoom to Selection Scaling

Alternatively, if you would like to customize the amount of whitespace being added to the drawing board, you can use the **Locator**.

The locator window is a viewpoint of the entire drawing board.

To display the locator:

1. **Click on the *Locator* menu item found under *View* > *Zoom***. The **Locator** window is displayed as shown in **Figure 2-8**.

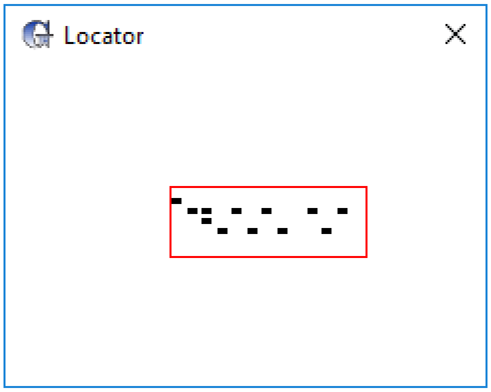


Figure 2-8 – The Locator Window

A smaller simplified representation of your layout is displayed in the window.

Also, shown in the locator window is a rectangular area, which corresponds to that portion of the total available drawing area currently displayed in drawing board.

To zoom in or out on a portion of the drawing board:

1. **Drag out an area on the *Locator* window**. The drag area should correspond to that portion of the total area you would like to view and will be represented by a red rectangle as shown in **Figure 2-6**. When you drag out a rectangle, the drawing board ***scale factor*** changes.

When the drag operation is complete, the drawing board will be updated, showing the selected area. If the selected area is smaller than the previous area, the effect will be to zoom in. Similarly, if the selected area is larger than the previous area, the effect will be to zoom out. If the selected area is offset to one side, the effect is to pan to that side (up, down, right, or left).

Figure 2-9 shows both the locator and the drawing board at two different scales. The top shows the default scale and the corresponding **Locator** window. When you drag out a smaller area in the locator window as shown in the bottom half of **Figure 2-9**, the scale factor is increased and the object icons appear larger.

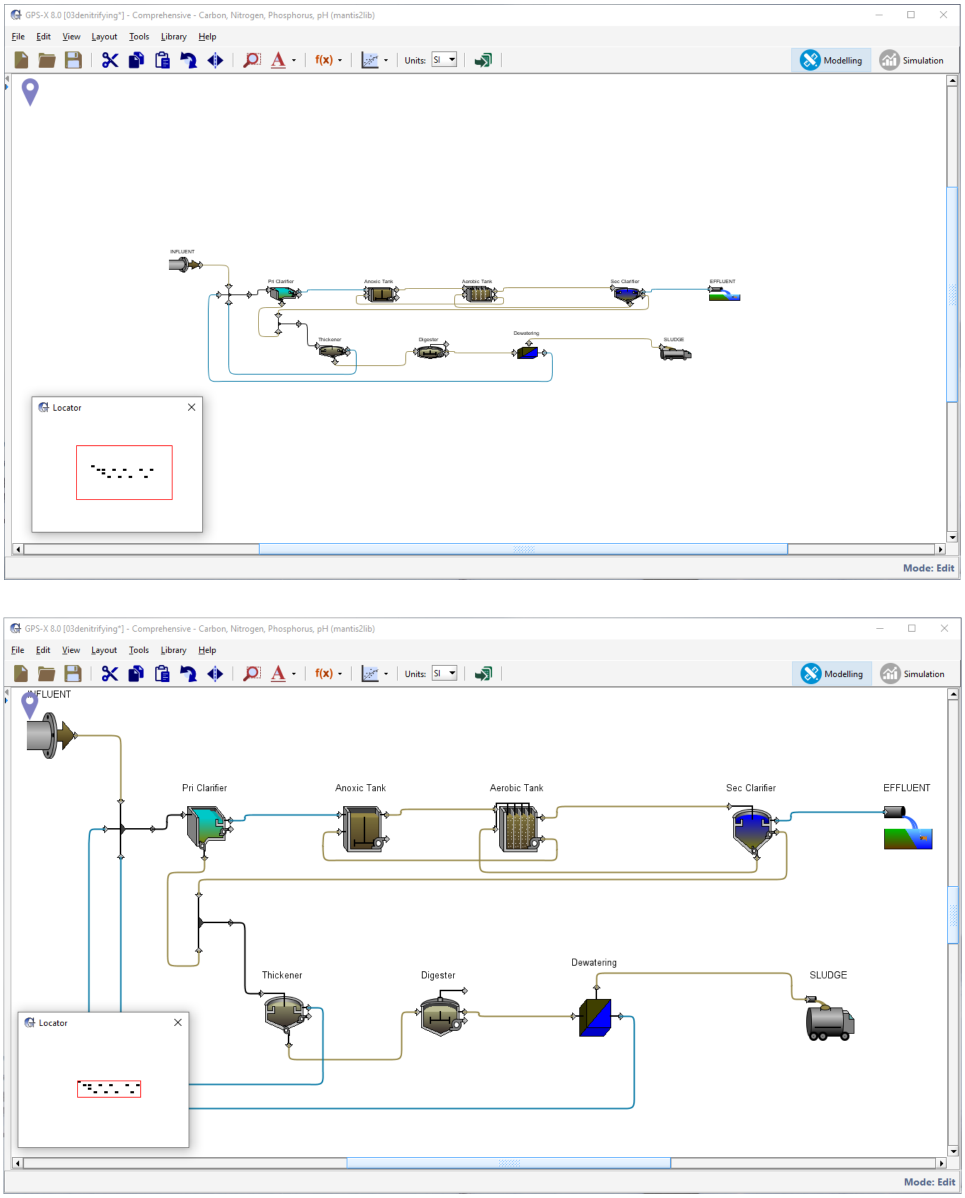


Figure 2-9 – Locator Window Scaling (2 views)

Layout History Database

The **Layout History** feature provides a means of storing the state of your layout through each stage of its development. Every time you save your layout, a copy is stored in a database for later retrieval.

To use this feature, select ***View* > *Preferences*** from the main menu bar and then select ***"Enable layout history"*** as shown in **Figure 2-10**.

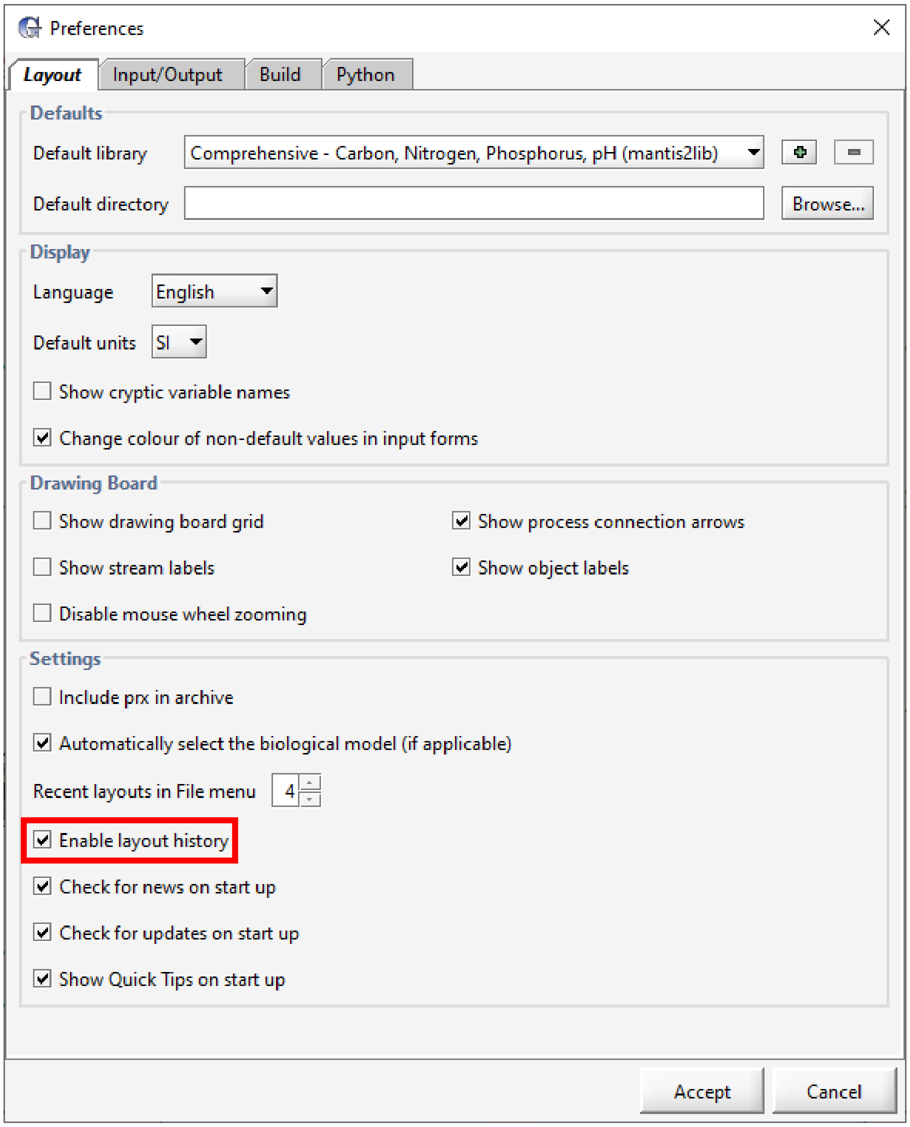


Figure 2-10 – Enable Layout History

When ***"Enable layout history"*** is first selected, a database for storing layouts is created in the user's login directory. The name of this database directory is ***.gpsxhistory***. When you select **Save** or **Save As…**, your layout will be backed up to the database. Also, any existing layout opened for the first time will be stored in the database.

To access your layout history, select ***File* > *History…*** as shown in **Figure 2-11**.

NOTE: This menu item is only accessible in **Modelling Mode**.

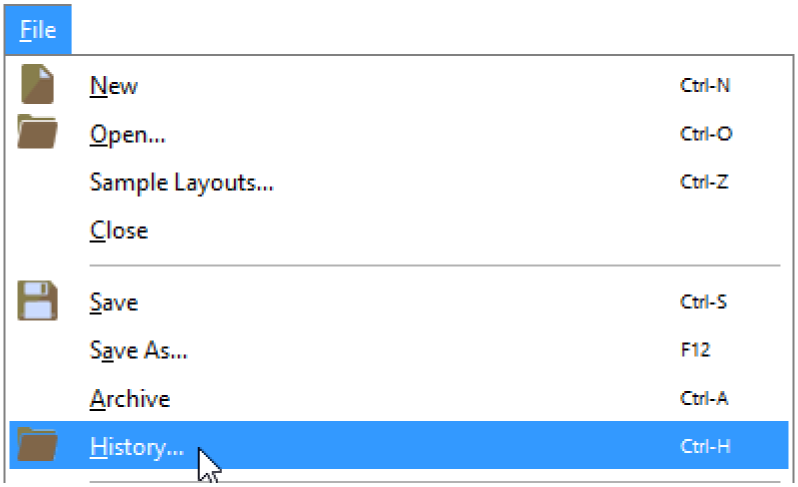


Figure 2-11 – Accessing the History Option

If your layout is already open in GPS-X you will see a dialog window showing its time stamped history as shown in **Figure 2-12**.

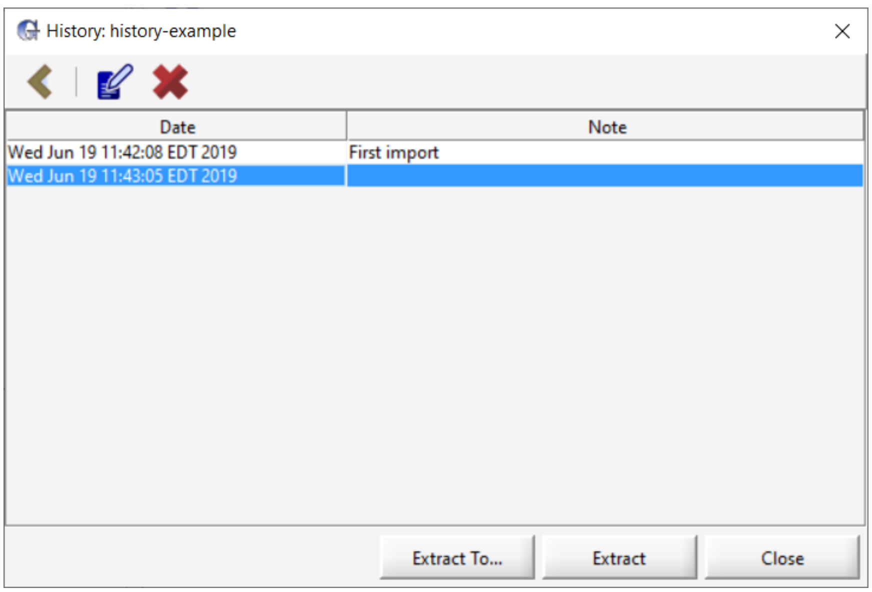


Figure 2-12 – History Dialog

Several options are presented to you in this dialog window.

(1) You can add a note at each time stamp to describe the layout at that point.
(2) You can select one or more entries to be deleted. By deleting entries that you are not interested in, you will save space on your hard disk and speed up database access.
(3) By selecting the desired time stamp entry and clicking **"Extract"** you will be asked if you want to overwrite the existing layout (which itself will have been backed up before being overwritten).

(4) Or select **"Extract To…"** if you want to place the layout in a directory different from the current one.

If no layout is open or you have clicked on **"Back to all Layouts"** in the dialog window as shown in **Figure 2-12**, you will see a different dialog window (see **Figure 2-13**) showing a list of all the layouts that have been stored in the database, including the directory paths to the layouts.

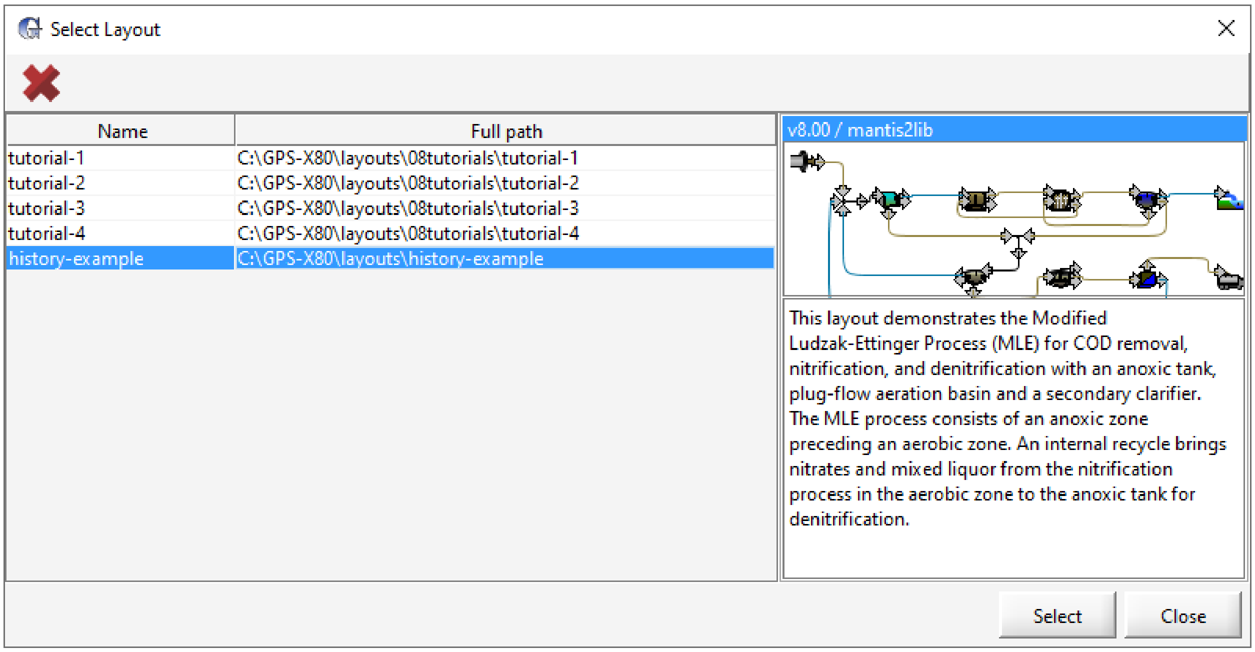


Figure 2-13 – History Layout Selection Dialog

After selecting one of the layouts and clicking **"Select"**, you will see the time stamped history dialog window for that layout.

Note that the database does not store any input data files (e.g. .dat or .xls) that you might have. The user is required to back up these files separately.

If you deselect ***"EnableLayout history"*** in ***Preferences***, the database is retained and will be available the next time you turn on this feature. You can back up the database directory (<User>/.gpsxhistory) as you would any other directory on your computer. You should deselect "***Save Layout History"*** in the ***Preferences…*** dialog first to be sure that the database is not active when doing the back up.

The history database is available to only one instance of GPS-X at a time. When the layout history feature is active, the history feature will remain greyed out in any additional instances of GPS-X that are opened. Consequently, layouts will not be saved to the database from this copy of GPS-X.

CHAPTER 3

Influent Advisor

The mathematical description of the influent wastewater that is fed to the plant model is the single most important aspect of a simulated system. Without significant consideration of the influent characterization, the plant model will be limited in its ability to predict the dynamic behavior of the plant. Hydromantis has developed a tool called ***Influent Advisor*** to help users navigate the influent models and achieve the influent characterization most consistent with the available measured data.

The influent model is dependent on the choice of a biological model chosen for the downstream processes. The reason for this is that the downstream biological models use different sets of state variables. GPS-X automatically determines which biological model has been chosen in the downstream processes and selects the appropriate influent model. This approach ensures that the influent data entered by the user is partitioned correctly. For instance, if a user is using ASM1 in a downstream biological process, then it is important that any biodegradable soluble COD be assigned to the *ss* state variable. Alternatively, if the user is using ASM2d (in which *ss* is not a state variable), then it is important that soluble substrate be assigned to *sf* or *slf* and not *ss.*

The influent advisor interface is shown in **Figure 3-1**.

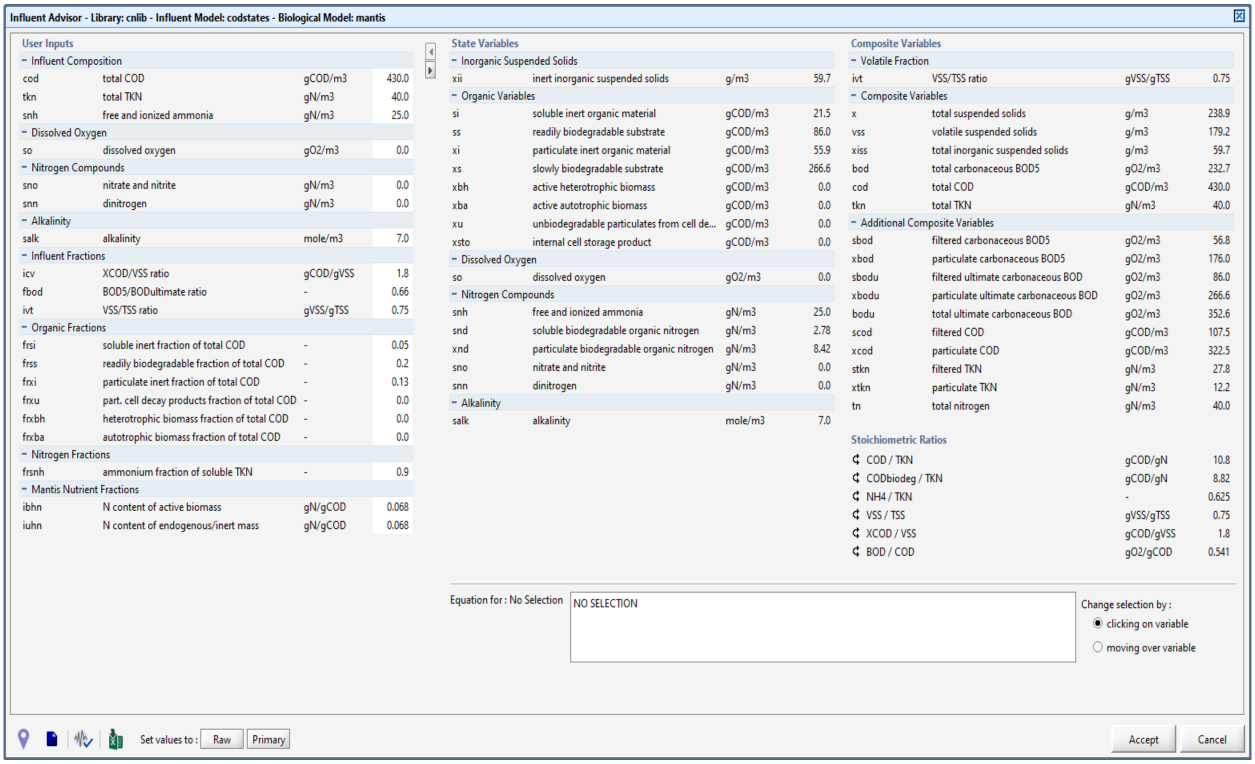


Figure 3-1 – Influent Advisor Example

The ***User Inputs*** column on the left contains the variables that can be directly modified by the user to characterize a given influent stream for the selected influent model.

The middle ***State Variables*** column contains a summary of the calculated states variables based on the concentrations and stoichiometry that have been entered in the left-hand column.

The third column located on the right lists the ***Composite Variables*** that have been calculated based on the specified user inputs and state variables. As the data is entered into the ***User Inputs*** column, the state and composite variable values are automatically updated accordingly.

To see how a value is calculated, click on it. The formula will be displayed in the *Equation for* box at the bottom of the screen, and the values used in the equation will be highlighted in the columns so that the applicable cells can be easily identified. If a negative value is calculated in either column, the number will turn red for easy identification. (**Figure 3-2**). Correcting problematic data is a matter of adjusting parameters in the ***User Inputs*** column to achieve non-negative values.

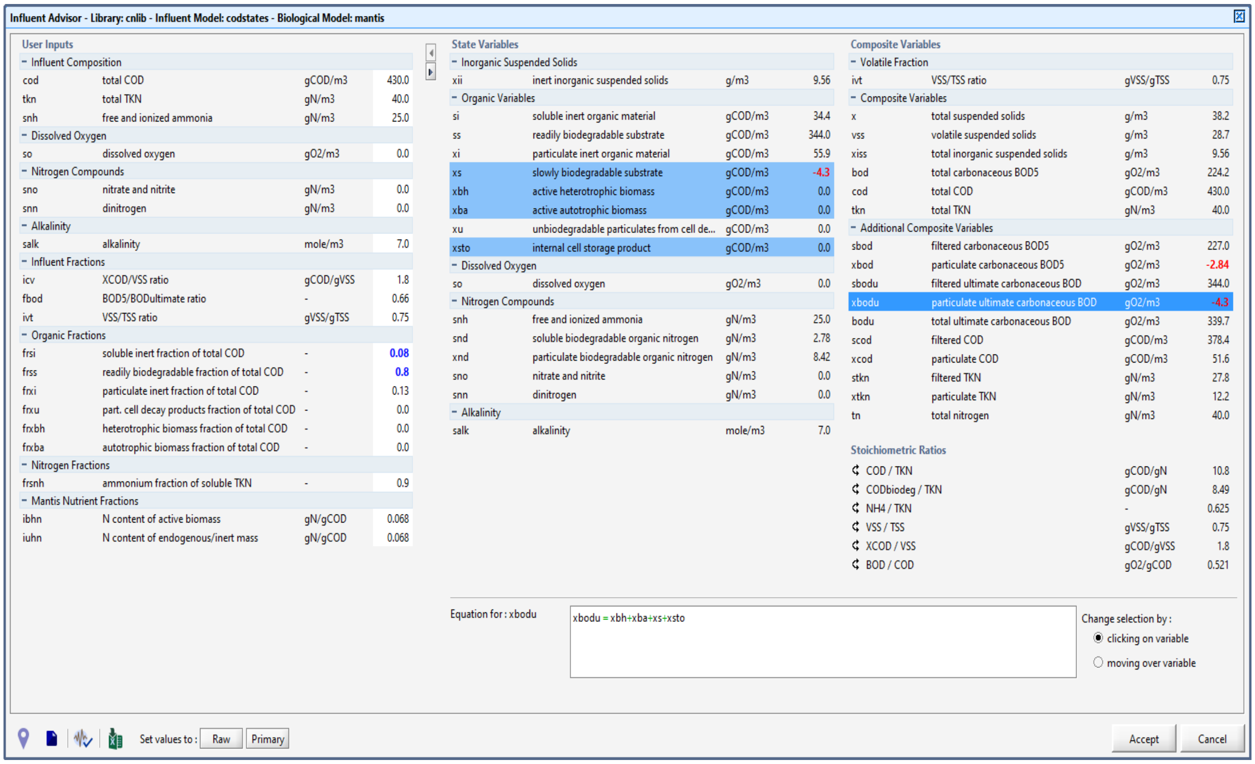


Figure 3-2 – Influent Advisor showing Highlighted Cells (Blue) and Negative Value Errors (Red)

The influent models used in GPS-X make certain assumptions about what data may or may not be available. For instance, no influent model uses both COD and BOD data, even though the data may be available and may provide important information about how that organic material is partitioned into the available state variables.

Influent Advisor helps overcome this shortcoming.

Take an example in which BOD and COD data is available and the **codstates** model is chosen for the influent object and the ***User Inputs*** cells can be filled in, including the available COD data. The user can then scroll to the right table and check the BOD values calculated based on the input data. If the BOD data is in agreement with the measured BOD data (assuming no negative values appear in any cells), then the user can be assured that the input data is consistent with the available data. If the BOD data is different from the measured BOD, then the unknown (or estimated) input data should be adjusted until acceptable agreement is achieved.

CHAPTER 4

Specifying Object Data

TYPES OF DATA

Influents, flow control points (i.e. splitters and combiners), and unit processes are characterized by the data that define them. Here, the term *data* is used in a generic sense to refer to the values of all attributes that uniquely define an object. Foremost among these *attributes* are the models which describe the input-output behavior of the object. Some models are simple and require few parameters. For example, a two-way flow-splitter model has a single parameter – the fraction of the flow going to one of the two outputs (the other is calculated). Other models are complex, such as the biological nutrient removal models used in reactor objects. The kinds of parameters needed for an object depend entirely on the kind of model selected for that object.

Attributes can be classified as either *numeric* or *text*. Numeric attributes include parameters like the real dimensions of the unit (width, area, etc.) and kinetic constants (maximum specific growth rate, decay constant, etc.). Most of the attributes in GPS-X are numeric. Text variables include the type of model used to describe the object, the type of controller to use in a model, etc. Text variables have a discrete number of '*values*'. For example, the plug flow tank reactor can have only one of several types of model – one of the ***asm1, asm2, asm3, newgeneral, mantis2,*** or ***mantis3*** models (not all of these models are available depending on which library you are using). Similarly, the ***controller type*** variable for the ***PID*** controller model can be ***P, PI,*** or ***PID***. When specifying a value for a text variable, GPS-X provides a list of options from which to make a selection.

Every object placed on the drawing board must be fully specified before a plant model can be prepared. GPS-X simplifies this using ***default*** models and parameter values. In practice, it is often found that some model parameters do not change significantly[1], or do so within a certain range. Incorporating default values for model parameters in GPS-X minimizes the amount of data entry that must be done before a working model can be built. However, you must ensure that these values are appropriate for the particular plant or process being modeled.

[1] Exceptions include the physical or operational parameters such as dimensions, operational mode, etc.

OBJECT MENUS

Object data are specified by opening the appropriate data entry form and either making a selection from a list of options or entering numeric values. Data entry forms are accessed from the **Process Data** menus, which are defined for each object.

To display the process data menu:

1. **Right click on any object on the drawing board**. The process data menu for that object will be displayed.

Figure 4-1 shows the process data menu for the circular secondary clarifier object.

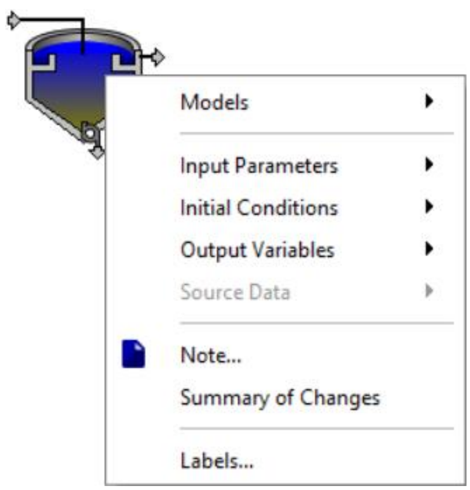


Figure 4-1 – Process Data Menu for a Circular Secondary Clarifier

Selecting a Model

In GPS-X, one of the most important attributes of an object is the type of model used to simulate the input-output behavior of that object. Because of this, the first step in specifying object data is to select the model to be used for each object. This is done by selecting the **Models** item from the process data menu.

To specify a model type:

1. **Pop up the *Process Data* menu** by right clicking on any object on the drawing board.

2. **Select the *Models* item**. This will cause a hierarchical menu to be displayed as shown in **Figure 4-2**.

3. **Click on the desired model**.

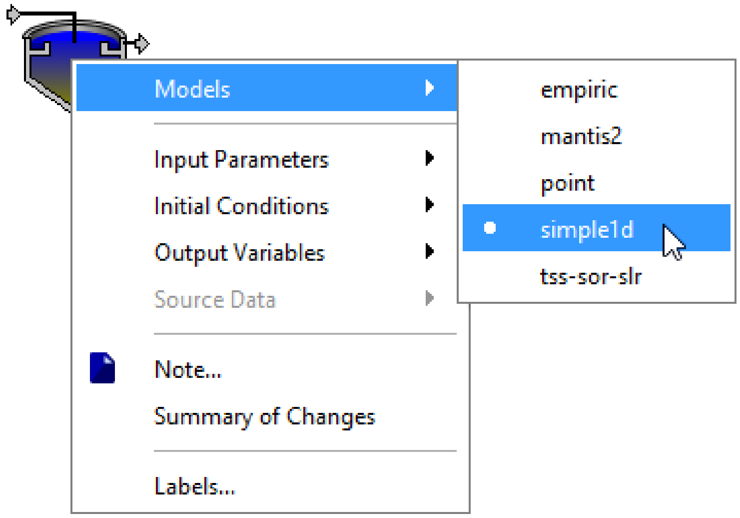


Figure 4-2 – Selecting a Model

GPS-X automatically selects the default model for you when you first place the object on the drawing board. If you wish to switch to a different model, you will lose any parameters entered into the data menus.

If you wish to change the default model so that GPS-X will always choose your desired model, edit the ***defaultmodelchoice.txt*** file in the subdirectory ***bin/gpsx/resources*** in the GPS-X installation directory.

Not all objects have more than one model. For example, splitter and combiner objects have a single, pre-set ***default*** model so you do not need to make a model selection to access the parameters for these objects.

Summary of Object Attributes

Once the model has been specified, you can access all of the object's attributes. These attributes differ depending on the model and type of object.

For detailed lists of object and model attributes, see the ***Technical Reference*** manual.

An object's attributes are divided up into several categories.

Input

Depending on the object type, the input variables are grouped into either:

(1) *Composition* and *Flow* if it's an influent object (see **Influent Objects**)
(2) or *Input Parameters* and *Initial Conditions* if it is any other object (see **Process Objects**).

The items under this group allow you to set the model characteristics in Modelling mode (see the **Data Entry Forms** section below) or select controller variables in Simulation mode (see CHAPTER 6 **Preparing Input Controls**).

Output

The output group allows you access to all the variables that can be displayed on graphs or various other output displays in Simulation mode.

Source Data

Allows you to specify (or remove) links to other unit processes where common data can be obtained. See the **Sourcing** section in this chapter for more details.

Note

Allows you to make any personal notes about this specific object on the drawing board. These notes are purely for your own reference. They do not affect the layout in any way.

Summary of Changes

Allows you to see a summary of all of the variables you have changed from their default GPS-X settings for a given object.

Labels

Allows you to view/edit the process and stream labels for this object (see the **Labels** section below).

DATA ENTRY FORMS

For a complete description of the data entry forms and parameters for each model, refer to the ***Technical Reference*** manual. The following is a general description of the forms.

Data entry forms are displayed when you select one of the menu items under an input grouping (ie. Composition, Flow, Input Parameters or Initial Conditions as described in the previous section).

Within each form, data are grouped into logical categories.

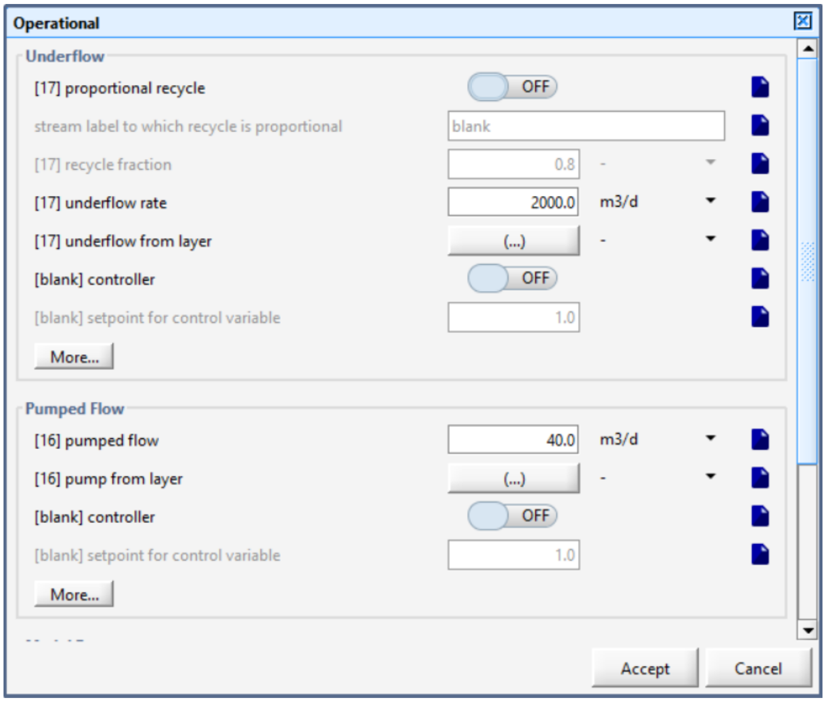


Figure 4-3 – Data Entry Form Example for a Secondary Clarifier

More…

In some cases, some of the less commonly adjusted parameters are grouped together on a separate data form, which is shown by the presence of a **More…** button.

Inactive Parameters

Some parameters will appear "greyed-out" (ie. inactive). These are parameters that are only relevant when another parameter has been activated or selected. For example, when the ***proportional recycle*** is set to **off**, the recycle parameters will be inactive. This functionality is also tied to the drop down menus.

Parameter Label

When the mouse is held over a parameter name, a tooltip appears which shows the descriptive and cryptic names of the parameter.

The purpose of this tooltip is twofold.

(1) One is to provide the descriptive name as some of the longer descriptive names may be cut-off in the data entry form due to space limitations.
(2) The second purpose is to show the cryptic name of the parameter. Knowing the cryptic name of a parameter is sometimes needed for setting up controller models, or for adding customized model code. For more information on the conventions used for setting cryptic parameter names, see the **Technical Reference** manual.

Entry Fields

To the right of the descriptive label for a parameter is an entry field for entering input. The entry field may be a data entry field, a text entry field, a drop-down menu from which a selection can be made, or an **ON/OFF** button. If the variable is an array, an **Array** button **[(…)]** is displayed. To access the individual array elements, click on the array button, and another form will be displayed for entering values in each element of the array.

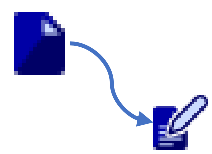
When values are entered that are different than the default value, the new value will be shown in **bolded blue text**. The default value can still be viewed by holding the mouse cursor over the entry field to display the tooltip.

Units

The unit of the input variable can be changed by clicking on the unit label.

Notes

To the right of the units is a button that allows you to add a note about each parameter.



Once notes have been made, the button's icon will change and the note will appear as a tooltip pop-up when the mouse is held over the button.

The notes function also supports the use of HTML tags.

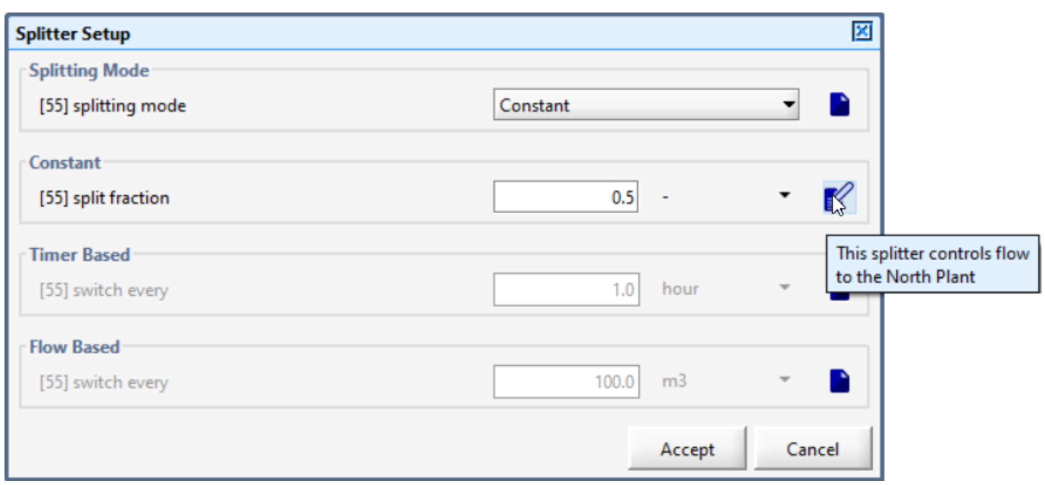


Figure 4-4 – Data Entry Form Example (with notes field in use)

INFLUENT OBJECTS

The composition, concentration, and flow characteristics of the influent are major factors determining the dynamic behavior in unit processes that make up the plant. GPS-X allows you to specify the type of flow or load pattern. You can specify a constant or sinusoidal flow or load pattern by entering only a few parameters in the ***Wastewater Influent*** object. You can enter a single diurnal pattern for GPS-X to repeat or you can enter your own flow and/or composition data and have GPS-X use these data for the influent. These attributes are entered by making selections from the **Composition** and **Flow** process data menu items.

Composition

The majority of an influent model's attributes are accessing through ***Composition*** item. These parameters have been further divided into categories depending on the specific type of influent (Continuous or Batch).

Influent Characterization accesses the Influent Advisor tool to aid in the setup of your influent object (see CHAPTER 3).

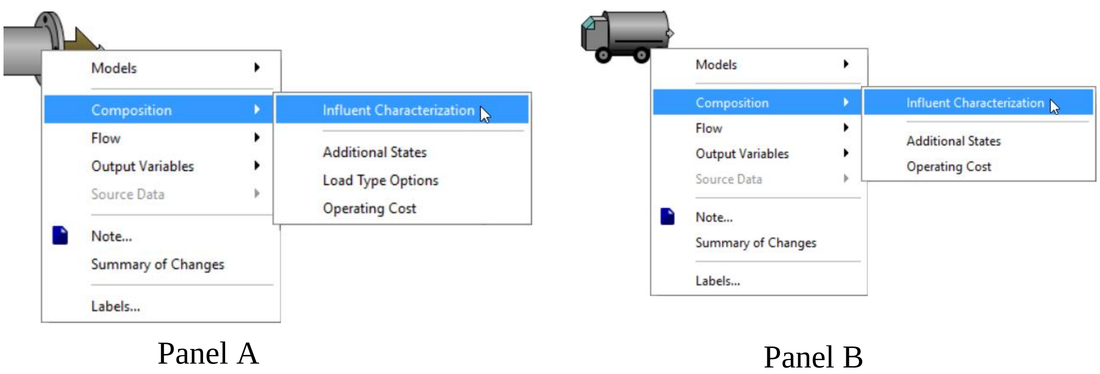


Figure 4-5 – Influent Object Composition Menus for (A) Continuous and (B) Batch

NOTE: Not all influent models have all the menu items displayed in Figure 4-5

Flow

The ***Flow*** > ***Flow Data*** item is used to enter a flow type and related data. The continuous wastewater influent object differs from the batch influent and chemical dosage objects in that it contains a specialized runoff model.

PROCESS OBJECTS

A variety of parameters are needed to specify a unit process; however, these can be categorized as either ***input parameter*** or ***initial condition*** values. ***Input parameters*** include physical, kinetic, operational and stoichiometric data; whereas, ***initial condition*** values refer to initial conditions for the model state variables including the volume and component concentrations.

Input Parameters

The majority of a model's attributes are accessed through the ***Input Parameters*** item. These parameters have been further divided into categories depending on the specific type of object.

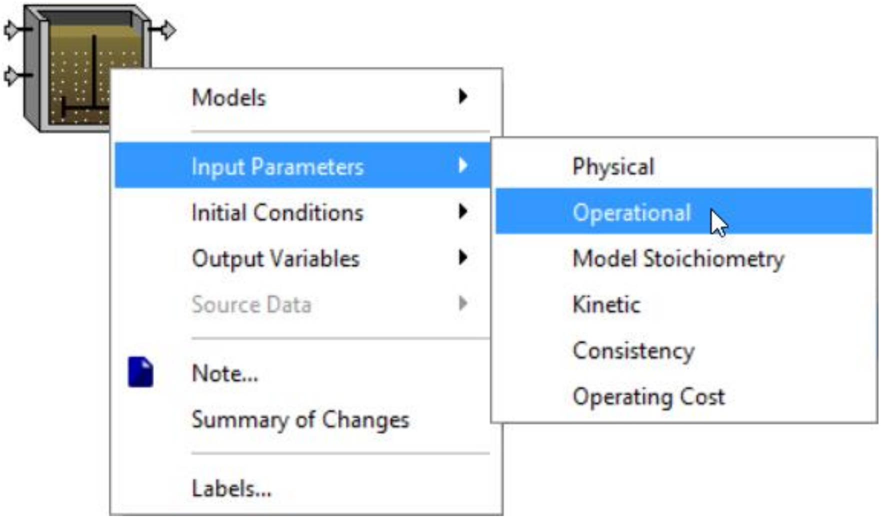


Figure 4-6 – Input Parameters Sub-Menu Example

Initial Conditions

Initial conditions are defined only for non-point process objects; that is, objects which have a volume. Point process objects include objects such as flow splitters and combiners, which are modeled as zero-volume control points. Point objects do not require initialization because there is no accumulation or utilization of material within the object. Point objects serve only to distribute or collect flows coming into the object.

In practice, initial conditions are rarely set manually unless data are available from previous simulations and you want to restore these conditions before re-starting the simulation, or if you are interested in starting the simulation with certain initial volumes (for example, with one or more tanks partially full). In the majority of cases, the default initial conditions – which are pre-defined for the model's default data – are used. For a given set of initial conditions, the usual procedure is to first run the steady-state solver to determine a valid steady-state, and then run a simulation. Alternatively, you can run the simulation for a period (i.e. 1 day), save the state variables, reinitialize the state variables and then conduct a simulation. See **Initial Conditions** in CHAPTER 8 for more information on setting the initial conditions for a simulation run.

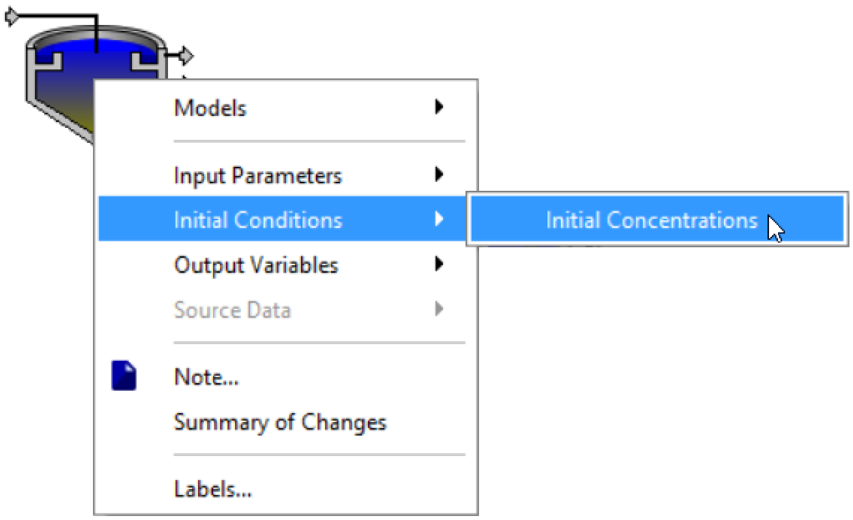


Figure 4-7 – Initial Conditions Sub-Menu Example

SUMMARY OF CHANGES

When constructing a model in GPS-X, the default settings for each variable provides a good starting point for most applications, but you will often need to modify these defaults to match your plant characterization and performance. For large models that are highly customized, it can be difficult to track all the changes that have been made to the GPS-X default settings. In Modelling Mode, the Summary of Changes menu item can be used to provide you with a quick summary of each variable in an object that the user has changed from its GPS-X default value. The Summary of Changes from the Default results for a wastewater influent object can be seen in **Figure 4-8**.

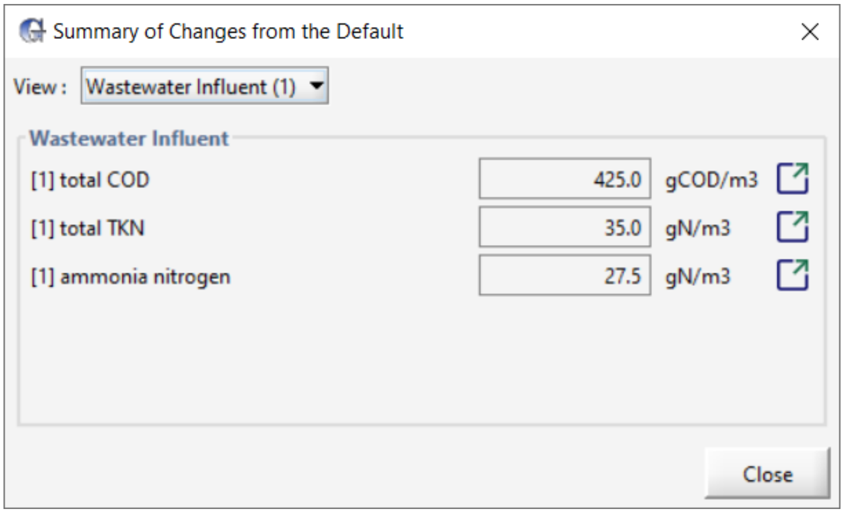


Figure 4-8 - Summary of Changes for an Influent Object

The name and current value of each modified variable for the selected object appears in the Summary of Changes output. Beside each variable value is a Go to location button which will take you to that variables data entry form where you can further manipulate the variable or reset its value to the GPS-X default value.

The View drop-down menu at the top of the output can be used to navigate between the objects on the layout. A Summary of Changes output will be available for each object that is currently placed on the drawing board.

LABELS

Among the important system-related attributes of every object in a GPS-X layout are the connection point labels for that object. When you drop an object icon on the drawing board, GPS-X assigns a numbered label to each connection point in that object. The label is used to generate model variable names, develop the model equations and manage the changes, which occur in each variable. These processes are transparent and need not interfere with the process analysis tasks you perform. In fact, a key advantage of GPS-X is its ability to hide lower levels of complexity so that you can concentrate efforts on process analysis and understanding.

In some cases, it is necessary or convenient to determine or possibly change the labels assigned to connection points on an object. To do this, you can use the ***Labels…*** item in the process data menu.

To change the labels on an object, complete the following steps:

1. **Right click on an object**. Select the **Labels…** item. This will cause the label dialog box to be displayed.

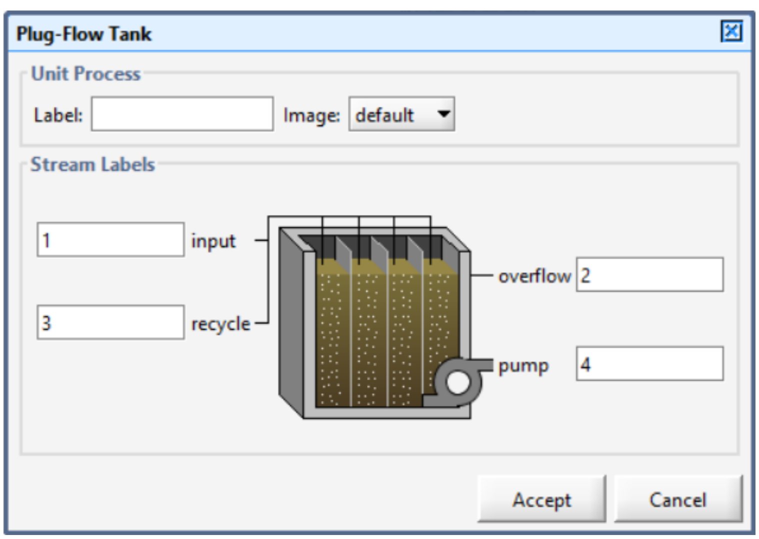


Figure 4-9 – Labels Dialog

2. **Enter any alphanumeric string at each connection point**. You can enter any string you like. The only constraint is that if the label consists of more than one alphanumeric character, the name must begin with a letter, not a number. For example, the labels ***a, 1,*** and ***a1*** are acceptable, but ***1a*** is not. GPS-X will not allow you to enter invalid strings (i.e. spaces are not allowed and certain characters cannot be used to start a string). The label can be up to 31 characters in length.

3. **Enter any alphanumeric string in the 'label' text box**. This is the process's label and is displayed above the object on the drawing board. It can also be useful for differentiating between objects when the output displays are created.

4. Click the **Accept** button to save these values. If you entered a label that is already in use, an error will be displayed and you won't be able to 'accept' the changes until the duplications are corrected.

Cryptic Variable Names

The connection point labels are used as flow stream identifiers and to generate variable names (referred to as 'cryptic' names) in the dynamic process model. Cryptic names are created by taking a default descriptor, such as ***'muh'*** for heterotrophic maximum specific growth rate, and appending the label to that descriptor.

For example, the cryptic variable containing the value of soluble BOD for an object's influent labeled ***'inf'*** is ***'sbodinf'***.

By convention, any variables internal to non-point objects take the output connection point label when constructing the variable name.

As an example, if a continuous flow stirred tank reactor has ***'140'*** as the label for the output connection point, the heterotrophic maximum specific growth rate (descriptor ***'muh'***) in the reactor would be named ***'muh140'***.

There are two ways to view the cryptic names.

(1) For individual parameters, position the mouse over the variable label of interest whereby a tooltip will appear with the descriptive label and the cryptic name separated by a back slash. See example in figure below.

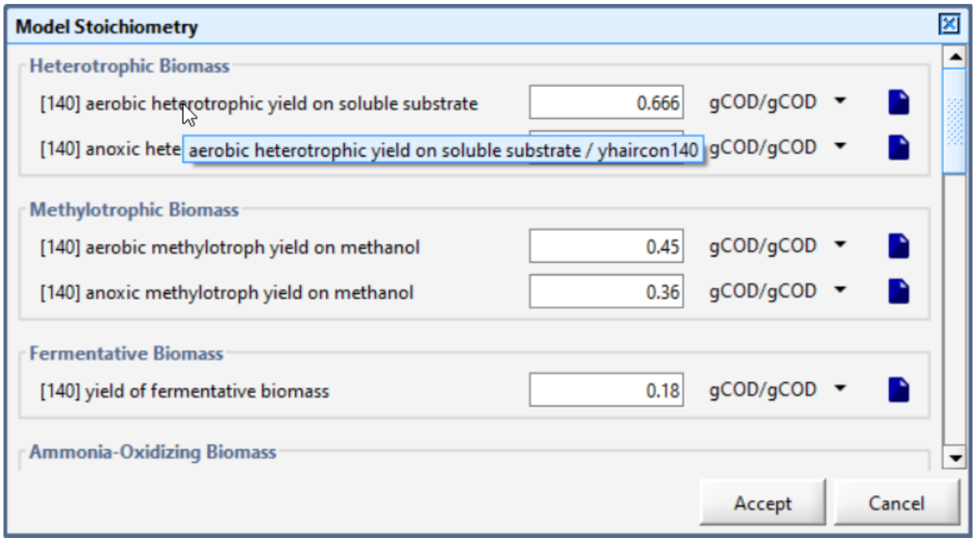


Figure 4-10 – Viewing the Cryptic Name via Tooltip

(2) Another way to view the cryptic names is by selecting the **Show cryptic variable names** option in ***View*** > ***Preferences*** > ***Layout*** tab under the **Display** group.

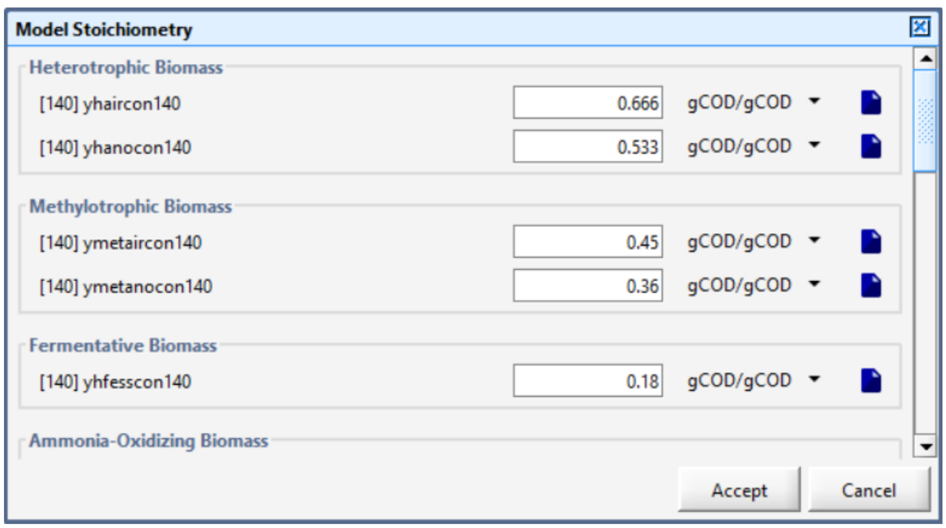


Figure 4-11 – Viewing the Cryptic Names via Preferences Setting

This will change it so that the cryptic name is shown in all data entry forms instead of the descriptive label. This can be confusing, so it is recommended that the cryptic option be used with care. The data entry label names will be refreshed each time you accept the changes to the Preference menu.

For more information on model variable names and conventions used in constructing these names, please see the ***Technical Reference.***

Displaying Labels on the Drawing Board

By default, labels are not displayed on the GPS-X drawing board[2]. If you would like to have them displayed, use the **Label** drop down menu on the **Toolbar** and choose either '***streams***', '***objects***' or both.

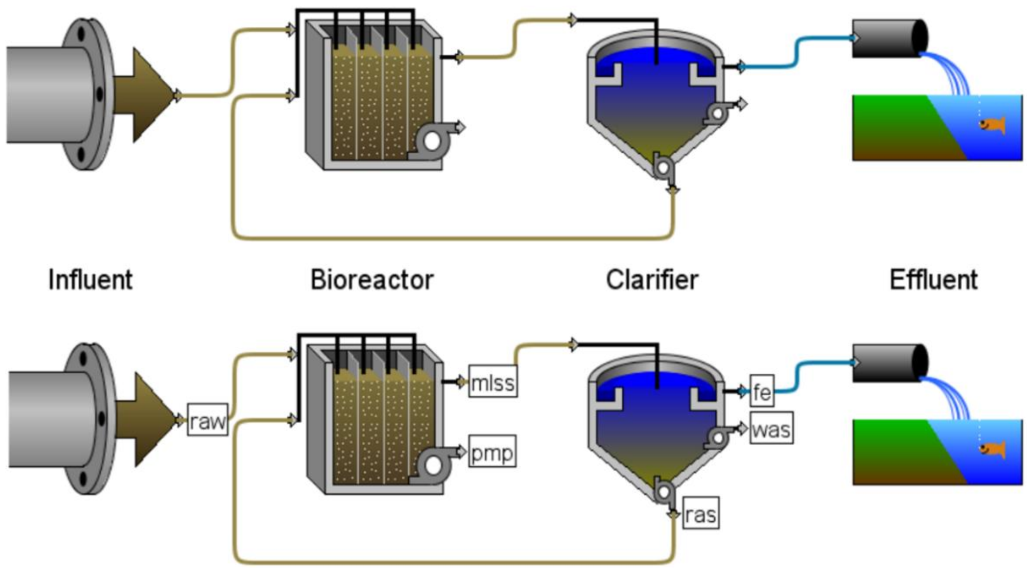


Figure 4-12 – Layout Before and After Displaying Process/Stream Labels

Labeling procedures in GPS-X are important when you need to uniquely identify an object and/or its streams. This is the case when object data are linked together as discussed in the ***Sourcing*** section of this chapter.

FIND

When constructing you are constructing a model, there may be times where you wish to modify, control or visualize a variable but you are unsure of which menu it is located in. If you know part of the variable's common name or have a cryptic variable name from a GPS-X output, the **Find** function can be used to locate the menu.

To open the **Find** window, go to *Edit* > *Find* to open the data entry window for the find functionality, as seen in **Figure 4-13**.

[2] This can be changed by checking the appropriate label checkbox on the **Layout** tab of the **Preferences** form.

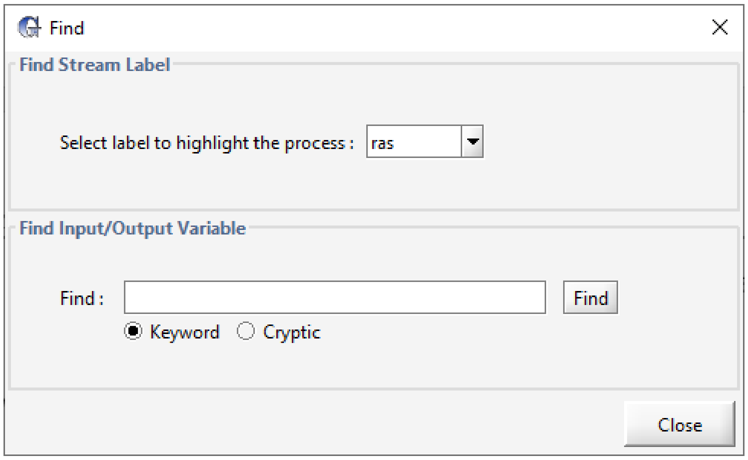


Figure 4-13 - Find Menu Entry Form

In the Find Stream Label section of the Find Window is a drop down which contains all of the stream labels on the layout. Selecting a stream label from this drop-down menu will highlight the object in the layout that the stream originates from. This can be used to identify the object that unfamiliar model outputs originate from.

The Find Input/Output Variable section of the Find menu is used to locate variables in GPS-X. If you are unsure of the variable you are looking for, keywords you expect to be in the variable name can be entered into the Find entry field with the Keyword toggle selected. For example, entering ‘underflow’ into the find entry and pressing the Find button will yield the results seen in **Figure 4-14**.

Using the find function will give you a list of all of the variables that contain the keyword ‘underflow’. The variables will be organized by both the object they are associated with on the GPS-X layout and if they are an input or an output variable. If you press the “Go To Location” next to the variable name it will take you to the menu where the variable is defined. If the variable is an input, you will be taken to a data entry menu, well an output variable will take you to a display variable menu.

Alternatively, if the cryptic variable name is known, the **Find** function can be used to can be used to find all variables whose cryptic variable name contain the known cryptic variable name. To search a cryptic variable name, enter it in the data entry field in the Find Input/Output Variable section of the Find menu and toggle on the Cryptic option below the entry field. Press the Find button and GPS-X will display all instances of the cryptic variable in the layout, formatted the same way as the Keyword find results.

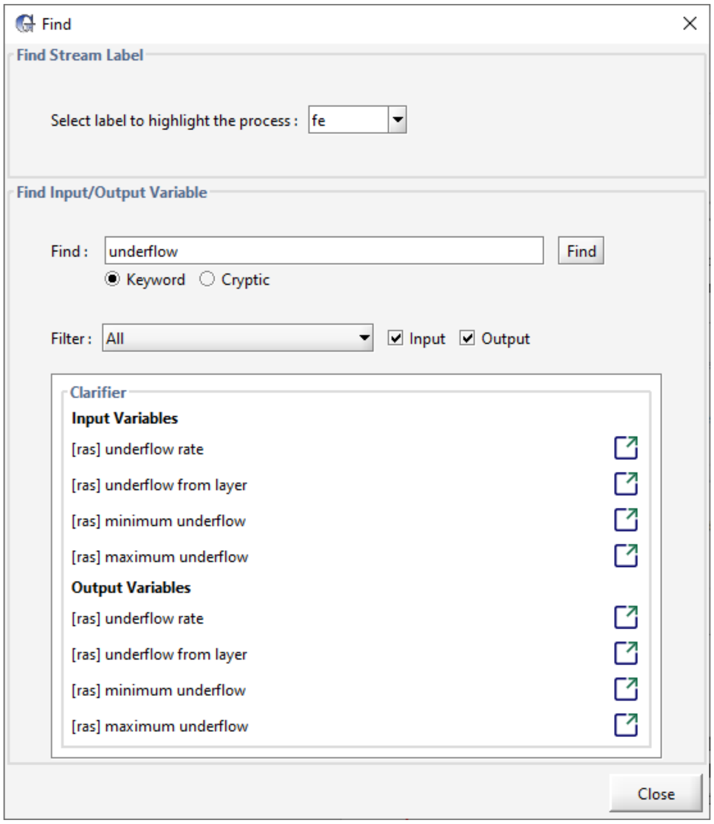


Figure 4-14 - Find Results for Underflow

SOURCING

Data entry is an important aspect of building and maintaining a dynamic process model. Once the model structure is in place, it is necessary to identify those parameters, which are most important and then determine how to adjust these to obtain the desired model behavior. With conventional modelling and simulation software, this task consumes a large fraction of the model development time, using resources that could be more profitably applied elsewhere. One time saving feature is specifying **Source** objects.

In GPS-X, attributes are linked to specific objects. This linkage mechanism is a simple way to reference objects to their data and makes it possible to link multiple objects to a single data set. When one object (a parent), becomes a ***source*** for a second object (a child), the latter inherits data from the former. This is the essence of the concept sourcing in GPS-X.

The real power of sourcing becomes evident when changes are made to object data, for example, due to model calibration, on-line data entry, etc. When objects are properly sourced, this change need only be made in one place – at the source object. If the sourcing feature was not used, these changes would have to be made for each appropriate object affected by the change.

NOTE: Not all data are inherited. Only the kinetic and stoichiometric parameters for reactor models and settling parameters for sedimentation models are inherited. The same model type must be selected for both objects before any sourcing links can be established.

Note the following constraints when sourcing:

- Sourcing of kinetic and stoichiometric parameters can only be established between objects with the same model type.
- Sourcing specifications cannot include inheritance loops, that is, you cannot specify a data chain of sourcing relations in which a parent object inherits from one of its child objects. It is not possible to determine a unique parent object in a sourcing chain forming a loop.
- Biological reaction unit processes can inherit kinetic and stoichiometric parameter values from any other biological reaction unit process with the same model.
- Sedimentation unit processes can inherit settling parameters from other sedimentation unit processes of the same kind. Secondary sedimentation and sequencing batch reactor objects can inherit from other secondary sedimentation or sequencing batch reactor. Sedimentation unit processes which have reactive models can inherit kinetic and stoichiometric parameter values from any other biological reaction unit process.

To specify a source object (parent) for another object (child) of the same type:

1. **Right-click on the child object**. The process data menu for that object will appear.
2. **Highlight the *Source Data* option** (**Figure 4-16**). If that option is disabled (greyed out) then that means that there are no potential data sources (ie. parents) currently on the layout.
3. **Select the parent object from the list**. The child will now automatically inherit data from the parent.

NOTE: A 'chain' icon will appear on the child object's image on the drawing board to easily identify which processes have been sourced (**Figure 4-17**).

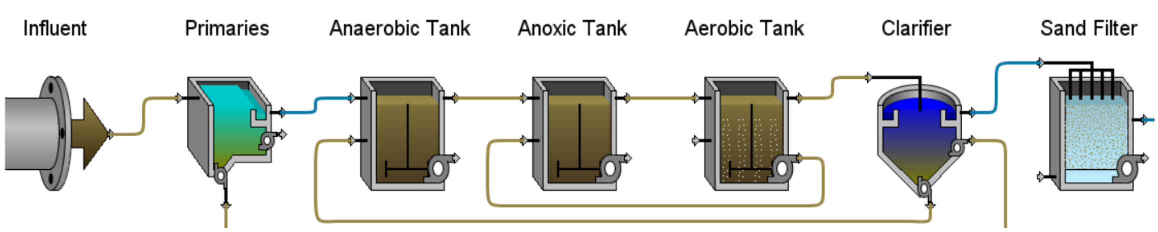


Figure 4-15 – Example Layout before adding Source Links

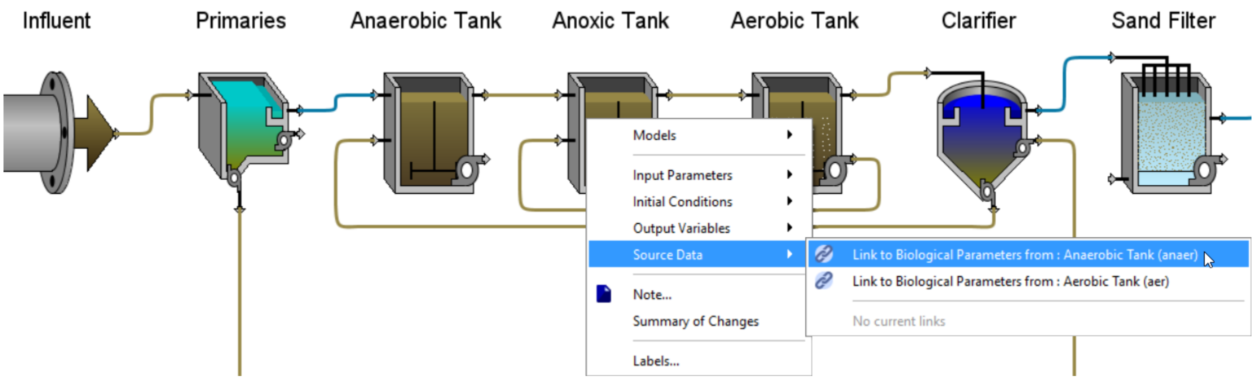


Figure 4-16 – Example Layout showing Source Data menu

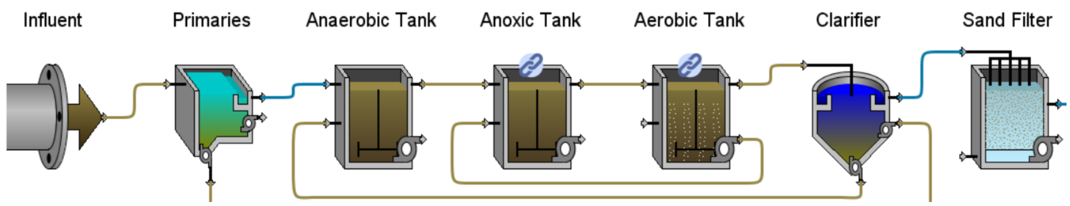


Figure 4-17 – Example Layout showing Source Linked Processes

To remove (or view) the current source of an object, complete the following steps:

1. **Right-click on the child object to display the process data menu.**

2. **Highlight the *Source Data* option.** The sub-menu will display other potential parents (if applicable) and below that is the option to remove the current parent.

GENERAL DATA

There are some data which are not exclusively related to influent, process or flow path objects. This includes operating system and simulator module set-up information, such as timing parameters, numerical integration options, and process environment parameters such as temperature. The data entry forms for this data are accessed from the Layout menu on the main menu bar. It can also be accessed by right-clicking on a blank area of the drawing board.

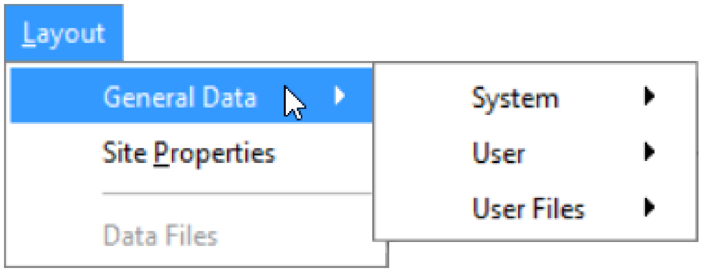


Figure 4-18 – General Data Menu

The **System** menu is used to gain access to the global simulation parameters, such as parameters related to the operation of the steady-state solver and the optimizer.

The **User** menu is used to gain access to the user-defined variables. This item is provided for advanced users to customize the GPS-X interface. The user can define any number of additional parameters or initialization and display variables and then access these here. For more information on customizing GPS-X see CHAPTER 11.

The **User Files** menu is used to define custom code and user-defined variables. This item is used to access the GPS-X user-customizable files, which give users the ability to define, use and display customized user code. For more information on customizing GPS-X see CHAPTER 11.

SITE PROPERTIES

The **Site Properties** item is merely a subset of the most commonly used data in the General Data category described above.

It allows users to customize the physical input parameters of the plant (Plant Site Properties tab) and the simulation date (Simulation Setup tab). Additional plant information notes can also be saved under the Plant Information tab.

CHAPTER 5

Defining Data

INTRODUCTION

You can create *secondary* variables that are calculated from existing model variables. Model variables are referred to as *primary* variables[3].

The **Define** feature is provided to make it easy for you to specify common secondary variables. You can add other calculated variables by modifying the model code directly. For more information on this topic, see CHAPTER 11.

You can specify daily and/or moving averages and mass flows for any variable in the model. This includes secondary variables that you may have defined previously, either in the code (CHAPTER 11) or with the define feature.

For example, with the define feature you can specify a secondary variable which contains the value of the moving average of the plant food-to-microorganism ratio.

Food-to-microorganism and solids retention time variables are special because normally they are calculated from primary variables at different locations in the plant.

For example, solids retention times can be calculated for a single treatment train, two or more treatment trains or the entire plant[4]. You can include the effluent suspended solids in calculation of the solids retention time or choose to ignore the effect of effluent solids on the calculated value. Similarly, you can evaluate the F/M ratio for a single reactor or multiple reactors and specify which unit process mass values should be used in the calculation. With the **Define** feature, you can specify how these variables are calculated for your plant.

The **Define** feature can be accessed through the ***Tools*** menu on the main menu bar or the ***Define*** button on the main toolbar.

[3] Another type of variable is a *composite* variable. Composite variables are calculated from the model state variables.

[4] In GPS-X, SRT is calculated on a total suspended solids (TSS) basis. The F/M ratio is calculated on a $CBOD_5$/VSS basis.

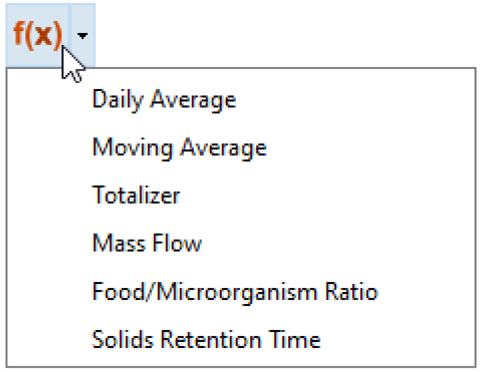


Figure 5-1 – Define Menu Options

The types of variables that can be defined with this feature include:

For any model variable:

- Daily average
- Moving average
- Totalizer
- Mass flow

Or any layout:

- Food/Microorganisms ratio
- Solids retention time

Use the define feature during set-up of the layout, and before building the simulation model. **You must always re-build the model after defining new variables**.

MOVING AVERAGES, MASS FLOWS AND TOTALIZER

Moving averages, mass flows and totalizers are often calculated for one or more variables in a treatment plant. The procedure for specifying these secondary variables is the same for all of them.

Moving Averages

Moving averages are data filters that smooth-out high frequency (rapid) changes in dynamic data so that it is easier to see overall trends.

Mass Flows

Mass flows are used in loading and material balance calculations.

Totalizer

This function will integrate the selected variable starting from time zero. The most typical application is integrating flow that is calculating the total liquid volume passed through the selected pipe during the duration of the simulation.

Concentrations totaled this way represent mass flow in a unit flow (1 m^3/d) over the duration of the simulation. If the total mass flow (g/d) is required, it is best to use the **Mass Flow** function in the **Define** menu.

Follow this procedure to specify these secondary variables:

1. **Click on the *Define* button** and select either Daily Average, Moving Average, Mass Flows or Totalizer from the list (**Figure 5-1**). A dialog window will appear as shown in **Figure 5-2**.

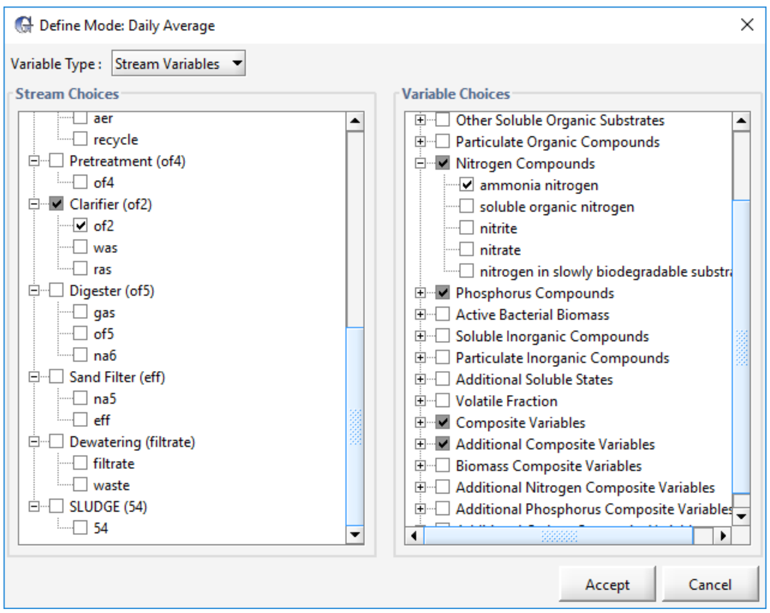


Figure 5-2 – Define Dialog for Daily Average

2. **Use the Variable Type drop-down box** to change the display to the desired type of variable.

3. **Select all of the locations (left side) and variables (right side) to include in the calculation.** Note that a checkbox with a grey background means that some, but not all of the items in that grouping have been selected.

4. **Click *Accept* to create the variable** and close the dialog window

The secondary variables are now specified and are valid model variables like any other in the model. This means that you can display the defined variable, query the simulator for its value and include it in the definition of other secondary variables. For example, you might want to calculate the mass flow of solids into a reactor or clarifier and, from this, a moving average of the mass flow.

Specifying Moving Average's Time Frame

After a moving average variable has been defined, the number of days to be used in the calculation can be specified in two different ways:

Through the Define dialog

(1) Open the ***Define*** dialog like you did in order to create the variable in the first place.
(2) Select "*Define Variables*" from the **Variable Type** drop-down box. A list of defined variables will be displayed.
(3) Find the variable of interest. Beside the variable will be an entry field where you can enter the number of days (**Figure 5-3**).
(4) Click ***"Accept"***

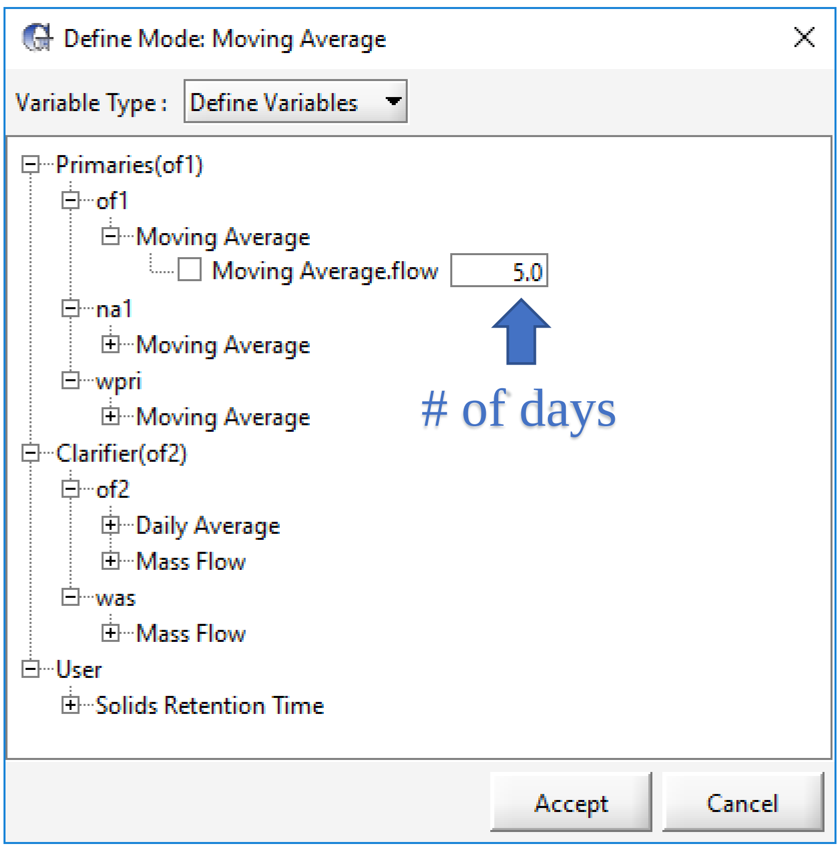


Figure 5-3 – Specifying Moving Average Time Frame (Method 1)

Through the Process Data menu

(1) Right-click on the process that contains the defined variable. The process data menu will appear.
(2) Select ***Output Variables* > *Defined Variables* > *Moving Average*.** See **Figure 5-4.** A dialog window with the defined variables will be displayed.

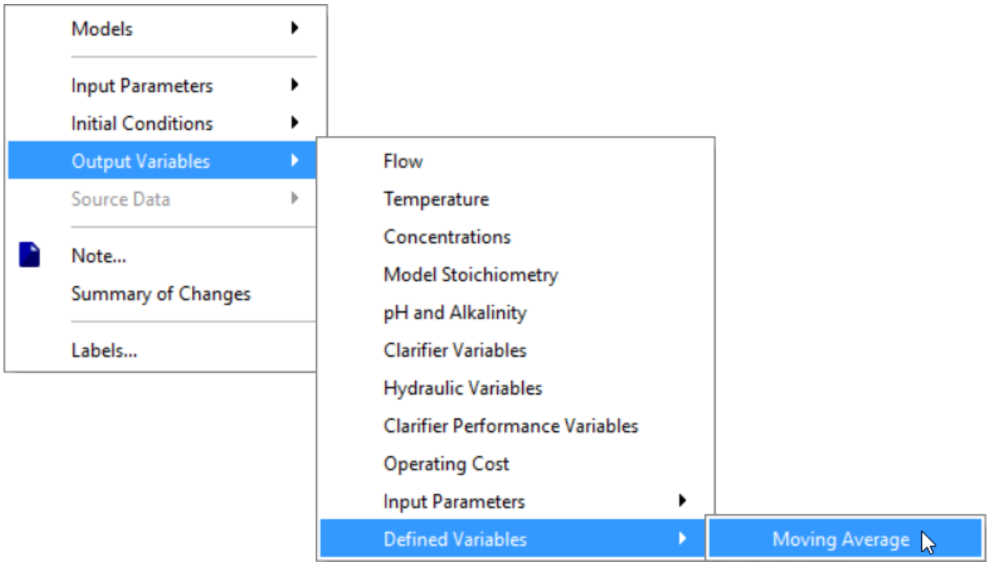


Figure 5-4 – Viewing Defined Variables

(3) Enter the desired value into the entry field (**Figure 5-5**)
(4) Hit ***"Accept"***.

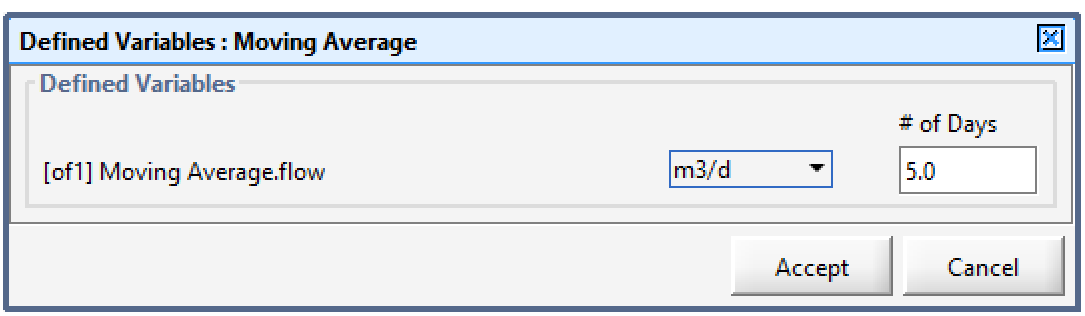


Figure 5-5 – Specifiying Moving Average Time Frame (Method 2)

The number of days that you set together with the communication interval specified for the simulation determines the number of points for calculation of the moving average. See CHAPTER 8 for information on simulation set-up including specification of the communication interval.

Remember that the communication interval defines the update frequency and the number of days value defines the overall time window used in calculating the moving average.

For the daily (moving) average the time window is 24 hours and is calculated exactly regardless of the simulation communication interval. To calculate the daily average, GPS-X uses data obtained at each integration time step rather than the output communication interval.

Any defined variable becomes a normal, calculated model variable. You can display the variable and use the variable in other calculations; however, as it is a secondary variable you cannot set its value before or during a simulation run. Remember that you must re-build the model after defining new variables to display the calculated values.

FOOD/MICROORGANISM (F/M) RATIO

The definition of food/microorganism ratio is often layout-dependent and, therefore, it is convenient to be able to define it differently for each layout you prepare. The procedure used to define the variable is interactive. Because it is based only on mass and flow measurements, the GPS-X define procedure has been designed to allow you to specify the form of the equation by simply clicking on the object or connection point containing the mass or flow value of interest.

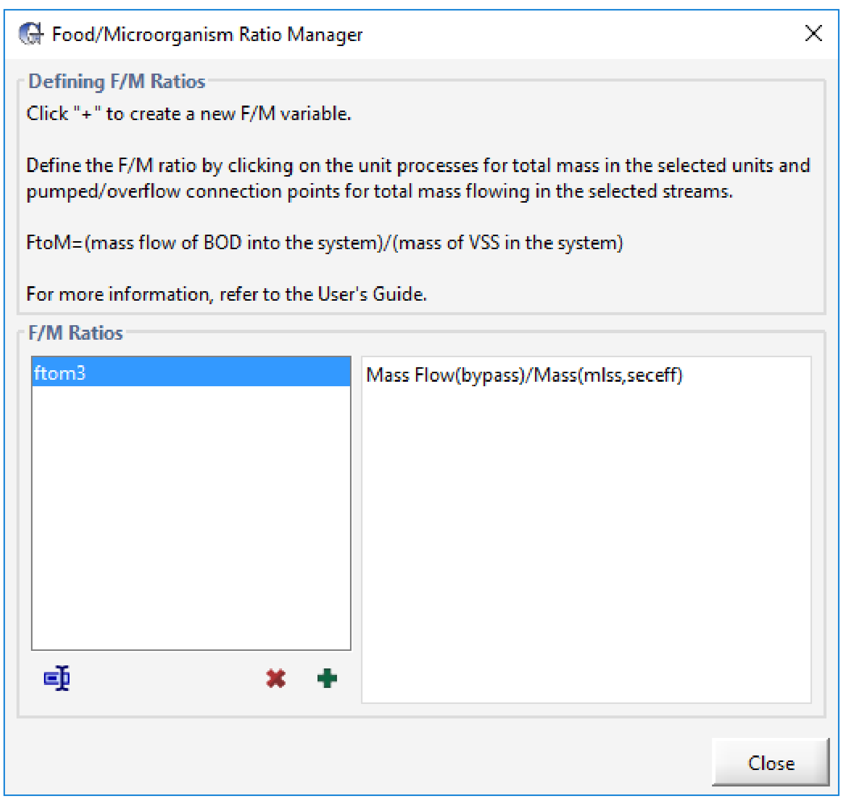


Figure 5-6 – Food/Microorganism Ratio Manager

To define a Food/Microorganism Ratio secondary variable, complete the following steps:

1. **Click on the *Define* button** and select the Food/Microorganism Ratio item. The F/M Ratio Manager dialog window will appear (**Figure 5-6**).

2. **Click on the "+" button to create a variable name for the new defined variable**. The text area on the right side of the dialog displays the mass and mass flow terms used in the equation to calculate the selected variable on the left. You interactively specify the full form of the equation by clicking on objects and object connection points.

3. **Click on a unit process on the drawing board (not a connection point) to add that unit's mass to the calculation of the mass term in the defining equation**. When you move the cursor over an applicable unit processes it will change the cursor to a hand. Clicking will add the object's identifier label[5] as an argument in the *Mass()* term. If you click the same object a second time, the object's identifier label is removed from the mass term.

4. **Click on connection points (not the object itself) corresponding to flow streams containing mass flow values of interest** to add that connection point's mass flow to the calculation of the mass flow term in the defining equation. When you move the cursor over an applicable connection point it will change the cursor to an arrow. Clicking will add the connection point's flow stream label as an argument in the *Mass Flow()* term. If you click the same point a second time, the connection point's stream label is removed from the mass flow term.

5. **After you have specified all appropriate mass and mass flow terms, click *Close*.**

SOLIDS RETENTION TIME (SRT)

The **SRT Manager** allows users to define multiple SRTs for a model layout. The SRT manger also allows the user to control sludge wastage via a SRT setpoint.

The SRT Manager can be used to define calculations for multiple SRTs if required. For example, it is possible to define aerobic SRT, anoxic SRT or anaerobic SRT by using the biomass in aerobic, anoxic, or anaerobic compartments respectively.

To define an SRT secondary variable, complete the following steps:

1. **Click on the *Define* button** and select the Solids Retention Time item. The SRT Manager dialog window will appear.

[5] By convention, this is the object's effluent flow stream level.

2. **Click on the "+" button to create a variable name for the new defined variable**. The text area on the right side of the dialog displays the mass and mass flow terms used in the equation to calculate the selected variable on the left. You interactively specify the full form of the equation by clicking on objects and object connection points.

3. **Click on a unit process on the drawing board (not a connection point) to add that unit's mass to the calculation of the mass term in the defining equation**. When you move the cursor over an applicable unit processes it will change the cursor to a hand. Clicking will add the object's identifier label[6] as an argument in the *Mass()* term. If you click the same object a second time, the object's identifier label is removed from the mass term.

4. **Click on connection points (not the object itself) corresponding to flow streams containing mass flow values of interest** to add that connection point's mass flow to the calculation of the mass flow term in the defining equation. When you move the cursor over an applicable connection point it will change the cursor to an arrow. Clicking will add the connection point's flow stream label as an argument in the *Mass Flow()* term. If you click the same point a second time, the connection point's stream label is removed from the mass flow term.

5. **The "Estimate WAS using selected SRT" checkbox** allows the user to estimate the waste flow rate for the selected SRT. Additional inputs like the SRT set point and min/max values for the calculated SRT controller pump flow are required.

6. **After you have specified all appropriate mass and mass flow terms, click *Close*.**

In the simulation mode, the values of the SRT can be visualized by dragging and dropping the SRT variables in the output window. The drag and drop action places both the instantaneous and dynamic SRT for visualization. Depending on the need, one or the other SRT can be removed from the outputs. In simulation mode, the SRT manager can also be used to turn off the WAS flow rate calculation based on SRT set point. The user can also change the set-point value in simulation mode.

[6] By convention, this is the object's effluent flow stream level.

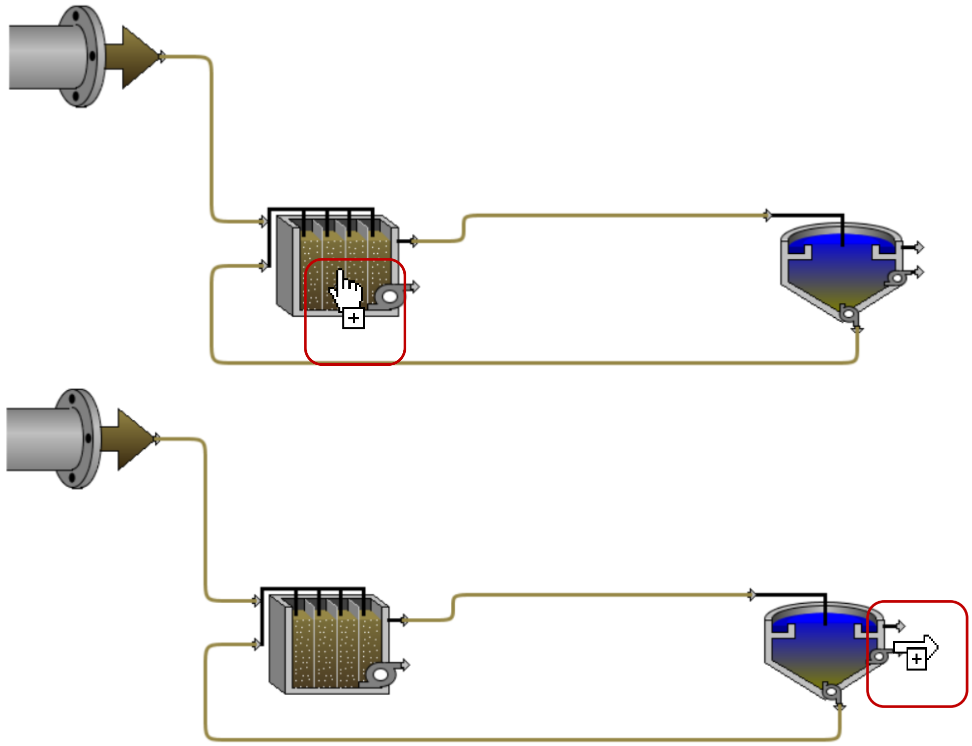


Figure 5-7 – Selecting Processes/Streams for Defined Variables

DYNAMIC SRT

Under dynamic conditions the solids retention time (SRT) is based on an age balance performed on the sludge in a manner similar to a typical mass balance. At steady-state the dynamic SRT is equivalent to the SRT. In dynamically changing conditions the dynamic SRT gives a better approximation of the true age of the biosolids in the system and can replace empirical smoothing methods (seven day moving average, etc.) routinely used to filter the sudden fluctuations in the instantaneous SRT calculation.

CHAPTER 6

Preparing Input Controls

WHAT IS AN INPUT CONTROL?

An input control is a user-defined graphical object that controls or modifies the value (continuous or discrete) of a model variable. This should not be confused with an *automatic process controller* which is used for maintaining a control variable at its set-point. The variable, which is linked to the control, can be any model independent variable. The effects of changes in the independent variable are observed by displaying selected *dependent* variables.

Any influent object flow or composition variable and any process object parameter or initialization variable, both continuous and discrete, can be linked to a control. These are the *independent* variables, which can potentially alter the dynamic behavior of *dependent* variables in the model. Dependent variables include the *state* variables, their *derivatives*, process rates, etc., any *composite* variables that are calculated from the states or their derivatives and any secondary variables that you may have defined. Dependent variables cannot be changed because they are the *response* variables calculated by the simulator's integration and calculation routines.

For example, for design purposes you may want to control the volume or surface area of a unit process (independent variable) and observe the effects on effluent water quality (dependent variable).

If you are interested in observing plant performance - measured by variation in effluent suspended solids, BOD_5 or ammonia nitrogen - at different solids retention times, you might want to investigate changes to the sludge wastage rate. Sensitivity analysis, which is a more structured investigation of relationships between variables in the model, requires that one or more dependent variables be observed with changes to the value of an independent variable within a certain range and at selected intervals.

Input controls are set up and used in the **Simulation Mode**.

There are three steps in creating and setting up an input control:

1. Create one or more blank input control tabs.

2. Locate the independent variable to be controlled, and drag it to the blank tab.

3. Access the properties to adjust the controller type and settings.

In **Simulation Mode**, the main window is divided up into three different regions; Controls, Outputs, and the Drawing Board (**Figure 6-1**).

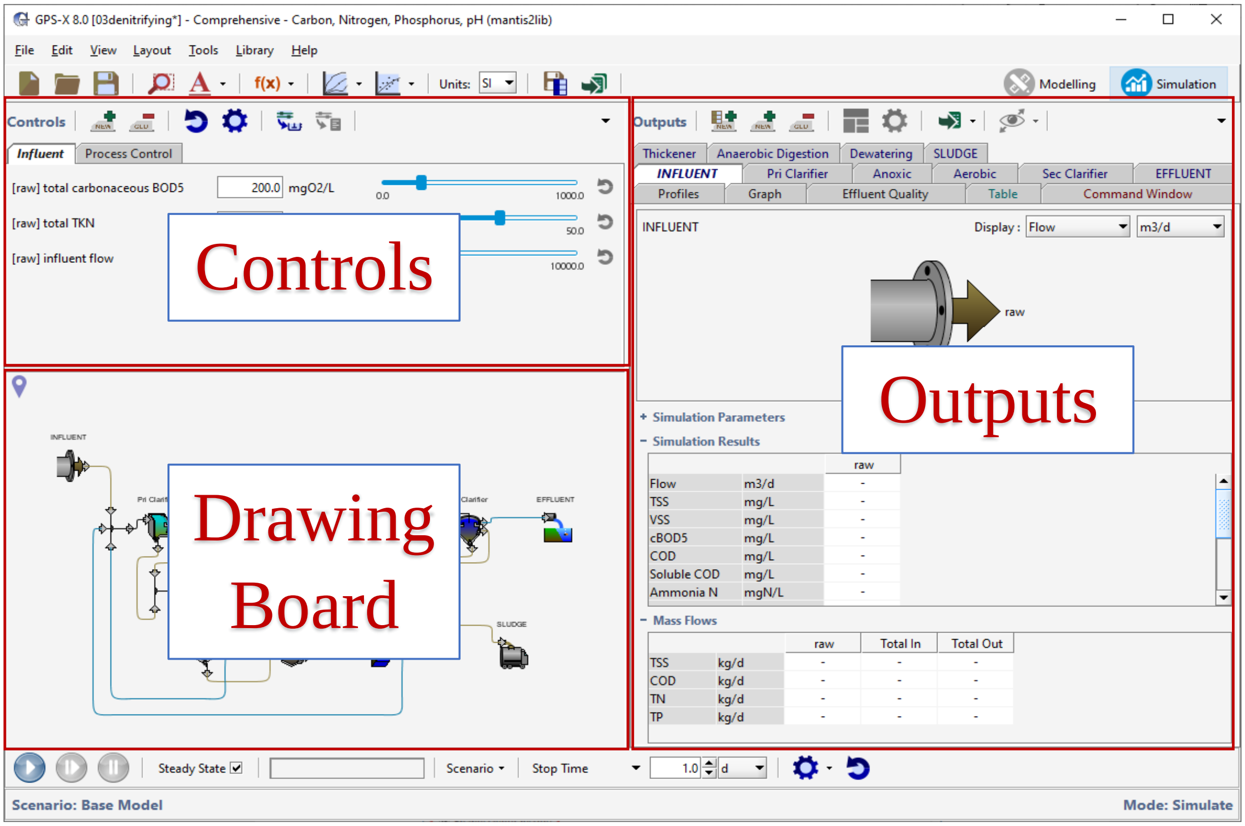


Figure 6-1 – Three Regions of Simulation Mode (Controls, Outputs, Drawing Board)

Input controls are displayed on one or more tabs. You can create as many tabs as you like and organize the controllers in whatever way is convenient. For example, you can place all the file input controls in a single tab and controls for the optimizer independent variables in another.

CONTROLS TOOLBAR

At the top of the Controls section of the main window is the Controls toolbar. This gives you access to many features pertaining exclusively to the input controllers

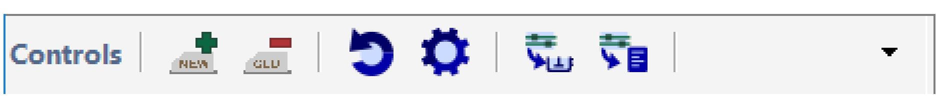


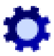
Figure 6-2 – Controls Toolbar

Here is a general overview of the buttons and their uses:

New Tab – Controllers can be grouped together into tabs. This button will create a new tab.

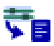
Delete Tab – This will delete the current tab and its contents.

Reset All Controls – This will reset all controllers on all tabs to their initial values.



Input Control Properties – See the **Input Controls Properties** section below.

Transfer Values to Layout – See the **Transfer Controller Values** section below.



Transfer Controls to Scenario – See the **Transfer Controller Values** section below.

Tab Listing – This displays a drop-down list of all the control tabs so that you can easily select the desired one. This can be useful when you have many tabs.

TYPES OF INPUT CONTROLS

There are two main types of input controllers:

- Direct (user interactive) controls
- Indirect (program-mediated) controls

Each of these control types is discussed in the next section.

Direct Controls

Direct controls are those that you can set up and change interactively as the simulation proceeds.

There are three types of direct controls that you can set up in GPS-X **(Figure 6-3)**:

1. **Slider Control:** The slider control sets the value of the independent variable and gives a visual indication of the setting and a text display of the current value. You must specify a minimum and maximum for the slider control.

2. **Increment Control:** This control consists of two buttons, which add or subtract a specified amount to the current value of the independent variable. You must specify the increment, minimum and maximum for this control. Three system-defined

controls that set the simulation stopping time, communication interval and delay make use of increment controls.

3. **Discrete Control:** One type of discrete control is an on/off (2-state) button, which allows you to activate or de-activate an independent variable. For multi-valued text variables, this control is displayed as a pull-down menu of mutually exclusive buttons. Pressing one of these buttons causes the others to pop-up

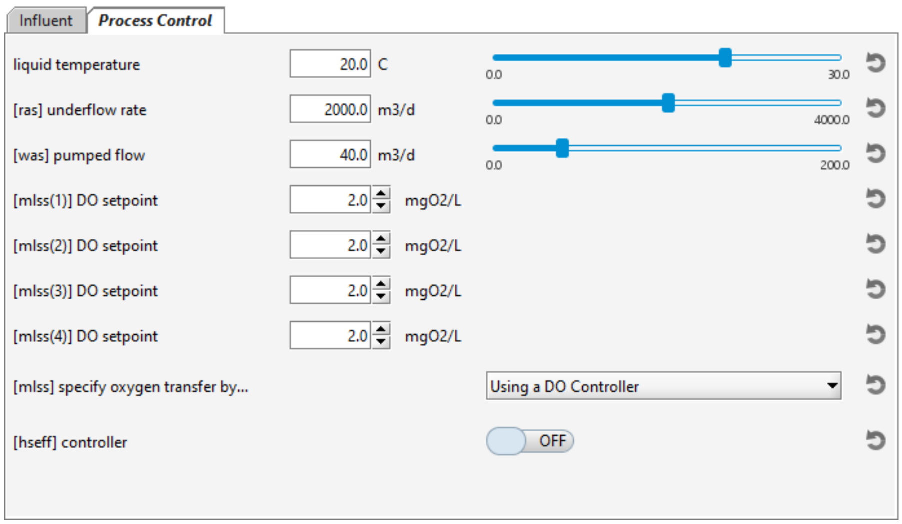


Figure 6-3 – Example of Different Types of Input Controls

With the slider or increment type controls, you can type in the value on the line to the left of the slider or buttons. When you press the Enter key after typing in the value, the new entry will take effect. In the case of the slider control you will see the slider pointer move to the new position. If you enter a value outside the bounds of the min/max, the number will change to red to alert you, but the value is still used as is.

The slider, up/down, ON/OFF and pull-down menu controls have a "reset" button located at the far right of the control. Pressing this button will reset the controller to the default value (taken from either the layout or the current scenario).

Indirect Controllers

The idea of a control extends beyond that of user-interactive control of an independent variable to include control of variable(s) by routines within GPS-X itself. These routines include:

- **File Input Control:** This function reads data from files. These data might be generated by other simulations, produced with spreadsheet or word processing programs, or collected from an actual plant. The data can be continuous, for example values of the influent flow rate, or they can be discrete, for example, ‘on’ or ‘off’ and ‘option 1’, ‘option 2’, etc. See the **Using File Input Controllers** section of this chapter for more information about setting up and using this feature.

- **Analyze Control:** These routines automatically vary the value of an independent variable. You specify the minimum, maximum and an increment and GPS-X runs the appropriate number of simulations to generate the sensitivity analysis results. There are additional input data requirements for sensitivity analysis. More information on setting up and performing sensitivity analysis is given in CHAPTER 9.

- **Optimize Control:** These routines calculate the optimal value of variables using specialized procedures to obtain targeted results. More information on setting up and performing an optimization is given in CHAPTER 10.

For each of these control types, you provide certain initial set-up information. GPS-X then controls, or sets, the value of the independent variable accordingly. The graphical objects displayed for each of these control types is a *non-interactive gauge*, which indicates the current value.

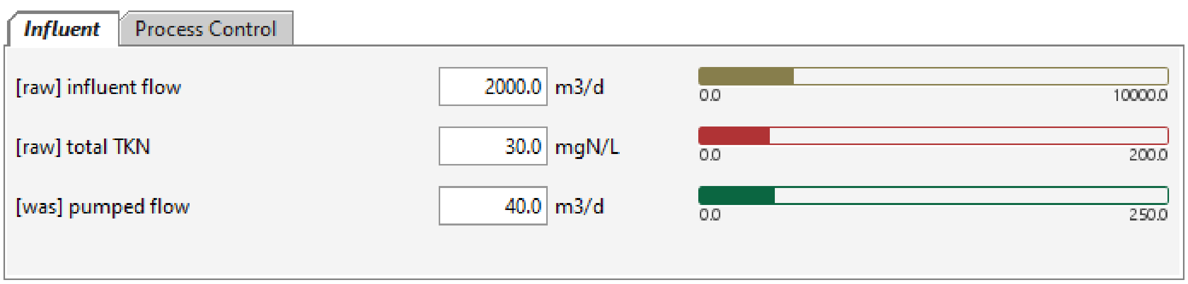


Figure 6-4 – Indirect Control Types – File Input (yellow), Analyze (red) and Optimize (green)

CREATING A CONTROL FROM AN INDEPENDENT VARIABLE

Specification of the independent variables for which you would like to prepare a control is done by dragging the desired parameter to an input control tab from a data entry menu (**Figure 6-5**).

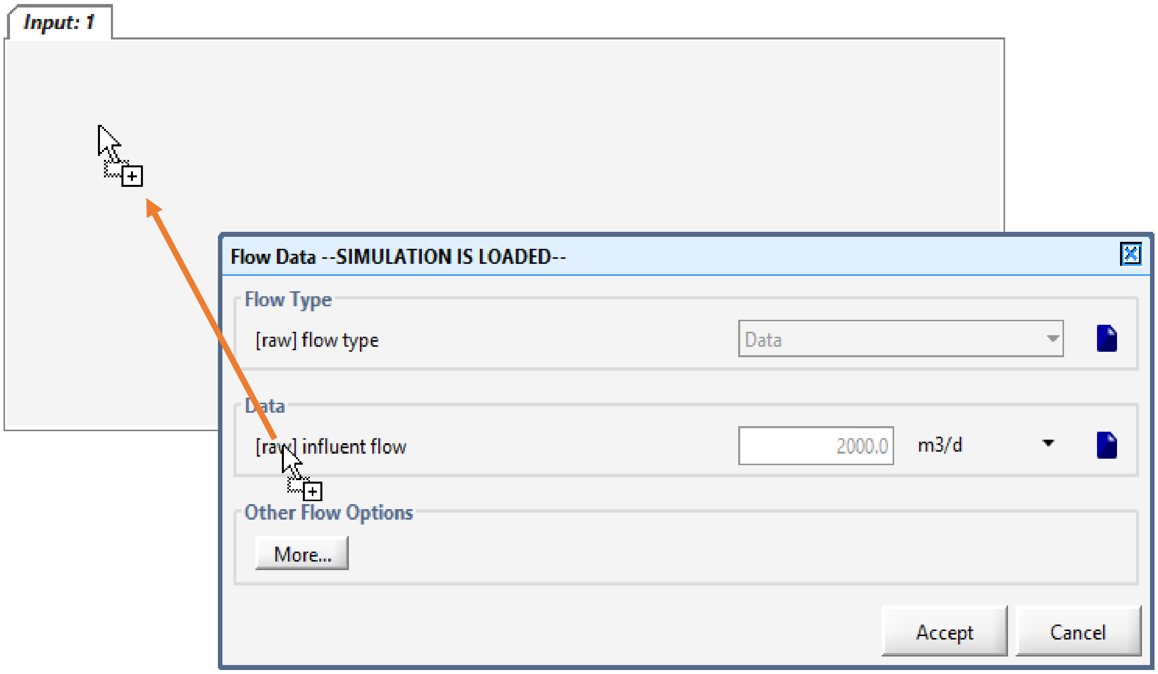


Figure 6-5 – Dragging Variable to Control Tab

To access the data entry menu, right-click on an object to display the process data menu for the object of interest and select the appropriate independent variable menu item as demonstrated in **Figure 6-6**. Data entry menus are accessed by selecting either the **Flow** or **Composition** menu items for influent objects or the **Input Parameters** or **Initial Conditions** menu items for other process objects.

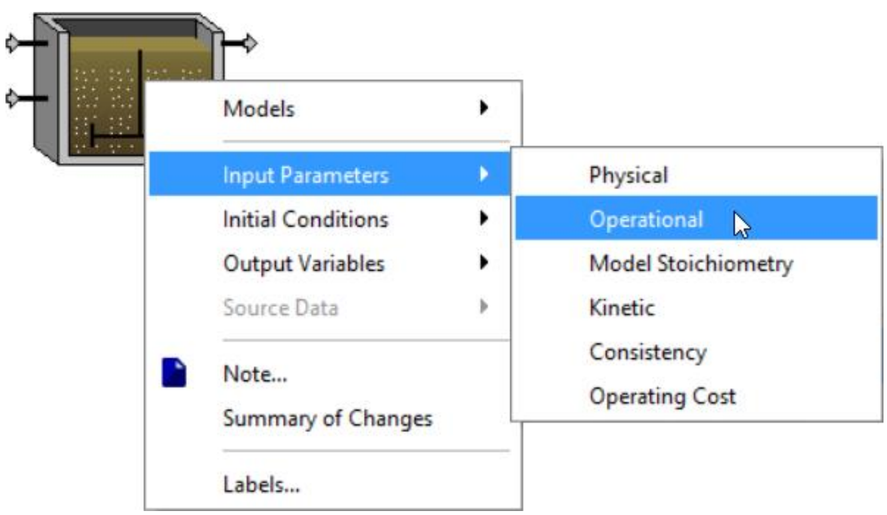


Figure 6-6 – Example of Process Data Menu

Some parameters in the data entry forms are unable to be dragged to an input control window because these variables cannot be changed as the simulation proceeds. These include variables that would require a recompilation of the model, such as the number of reactors in series in a plug flow reactor.

It is important to differentiate between setting up input controls and setting up output display graphs as described in the next chapter. The procedure for both is similar; however, the selection of display variables is made only from the **Output Variables** menu item for a process object.

Quick Tip: You can quickly create new input control tab by dragging a parameter (from a menu or another input control window) to the blank area to the right of existing tabs. By doing so, a new tab will be created and the parameter assigned to it. This eliminates the need for clicking on the "**New Tab"** button first.

INPUT CONTROLS PROPERTIES

Once you have created one or more control tabs and filled them with parameters to be used as interactive controls, you can then specify the properties of the controls themselves.

Clicking the **Input Controls Properties…** button (**Figure 6-7**) will bring up the **Control Properties** dialog.

Figure 6-7 – Accessing Input Control Properties

In this dialog window, you can specify the units, ranges, and types of controllers for each input control parameter, as shown in **Figure 6-8**.

There are several different options for the type of controller, including:

1. Slider
2. Up/Down (increment)
3. On/Off (discrete variables only)
4. Analyze (Step & Monte Carlo)
5. Optimize
6. File Input
7. Database

Note: Some options may not be available depending on the specific variable

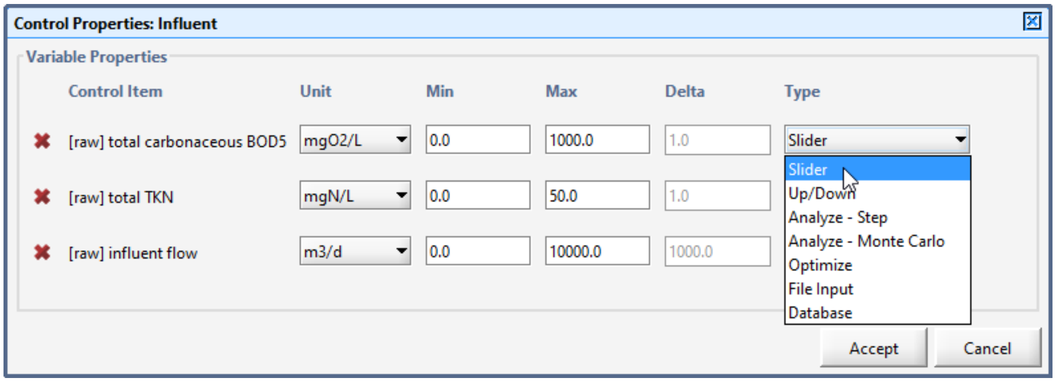


Figure 6-8 – Input Control Properties showing Control Type Options

The interpretation of minimum (**Min**), maximum (**Max**) and increment (**Delta**) values is different for each type of control as described below.

- **Slider Control:** The minimum and maximum values define the total range of the slider. The resolution of the slider is 1/100th of the range calculated as maximum value - minimum value. The value of delta is ignored.

- **Up/Down (increment) Control:** The minimum and maximum values define the range over which the variable can be incremented. The value of delta is taken as the increment value.

- **On/Off (discrete) Control:** For this control, there are discrete choices (two or more). Values for minimum, maximum, and delta are ignored.

- **Analyze Control:** The minimum and maximum values define the limits of the analysis. For more information about analyze control, see CHAPTER 9.

    - In a step sensitivity analysis, the independent variable varies from the minimum value to the maximum value in increments of delta.

    - In a Monte Carlo analysis, the independent variable varies randomly within the range following the defined probability function

- **Optimize Control:** The minimum and maximum values are used as constraints in the optimization procedure. The optimizer will not set the independent variable to a value less than the minimum nor greater than the maximum. The delta value is ignored. For more information about optimize control, see CHAPTER 10.

- **File Input Control:** The minimum and maximum values are used to filter the input data. When a datum is read, it is compared with these values. If the input datum is less than the minimum, then it is set equal to the minimum value. If the input datum is greater than the maximum, then it is set equal to the maximum value. The delta value is ignored. For more information about file input controls, see the **Using File Input Controllers** section of this chapter**.**

- **Database Control:** Similar to the file input control, this control reads a time series of data from a database (such as MySQL). The minimum and maximum values are used to filter the input data. When a datum is read, it is compared with these values. If the input datum is less than the minimum, then it is set equal to the minimum value. If the input datum is greater than the maximum, then it is set equal to the maximum value. The delta value is ignored. Use of the database input control is limited to users with the Advanced Online Features option activated in their license. For further information, please contact Hydromantis.

RENAMING AN INPUT CONTROL TAB

The name of an input control tab can be changed by simply double-clicking on the tab name, changing the text, and pressing 'Enter.'

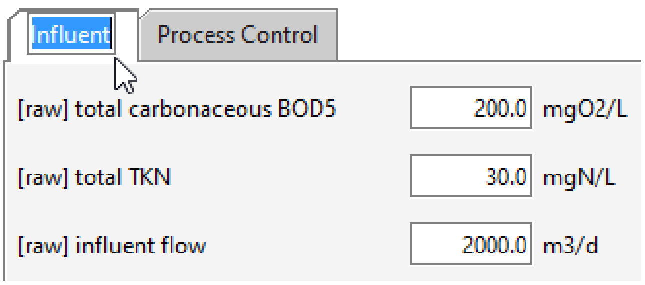


Figure 6-9 – Renaming a Control Tab (Method 1)

Alternatively, right-clicking anywhere on the tab (other than parameter names) and selecting "**Rename Tab"** will bring up the window shown below:

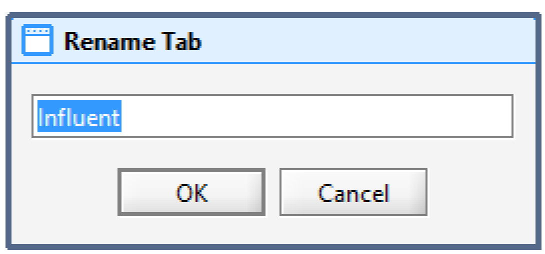


Figure 6-10 – Renaming a Control Tab (Method 2)

REMOVING A CONTROL

To remove a variable from an **Input Control** tab, right click on the parameter name, and select **"Remove Input Control"**, as shown in **Figure 6-11**.

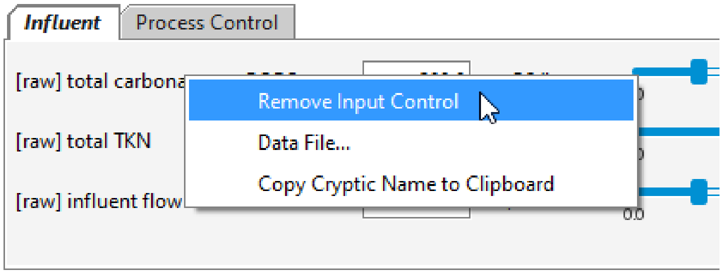


Figure 6-11 – Removing an Input Control (Method 1)

You can also remove the controller by accessing the "Input Control Properties…" dialog window and clicking on the red 'x' beside the variable (**Figure 6-12**).

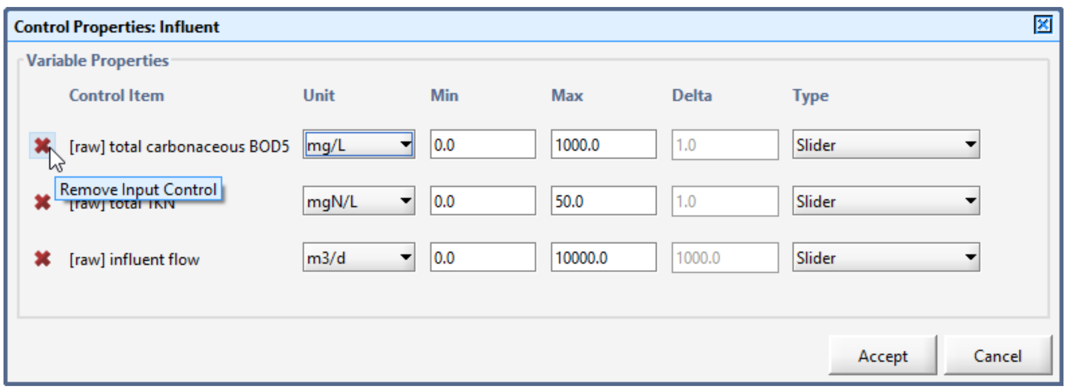


Figure 6-12 – Removing an Input Control (Method 2)

TRANSFER CONTROLLER VALUES

There are times when you may have used the controllers to fine-tune a calibration of your model and once you are satisfied with the results, you'd like to use those values as the default values for either your layout or a scenario (for information on scenarios, see the section **Using Scenarios** in CHAPTER 8).

To accomplish this, we have added two buttons to the Controls toolbar to simplify this task.

Transfer Values to Layout

This feature will give you the option of selecting (from the current tab) which control values you would like to transfer and use as the default values of your layout. Transferring values to the layout will cause the layout to be saved and the model to be rebuilt with the new values.

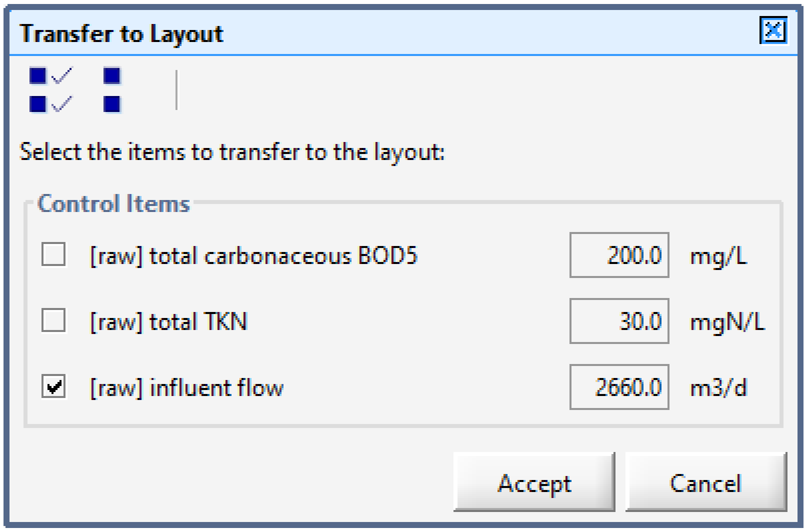


Figure 6-13 – Transfer to Layout

Transfer Controls to Scenario

This feature is only enabled if a scenario has been selected (see **Using Scenarios**). It will give you the option of selecting (from the current tab) which control values you would like to transfer and use in the scenario. As opposed to transferring the values to the layout, this action does not require for the model to be rebuilt.

USING FILE INPUT CONTROLLERS

In many cases, it is desirable to use data generated outside of GPS-X as input to the simulation model.

For example, it might be convenient to use actual plant influent flow and composition data as input to the model to observe the difference in response between the plant effluent data and model predictions. Operational reports, such as those recording a step-feed strategy, might be used to set up a file containing instructions on how to change the model to mimic the operation in the real plant.

The **File Input** control type reads data directly from a properly formatted ASCII or Excel file.

There are two methods for preparing data for a **File Input** controller: manually preparing data files, and using the GPS-X data file editor.

Manually Preparing Data Files

Any number of data files (in standard ASCII text or Excel spreadsheets) can be read and used as model inputs (input excitations or driving functions) or for display in GPS-X.

While the files can reside anywhere on your computer, it is suggested that you place them in the same directory as the layout files or a subdirectory of that location.

ASCII Text Files

For ASCII text files, you'll need to know the cryptic variable name and the time-stamped data values. The file must be saved with the extension .dat or .txt.

The format for this file is as follows:

```
t<delim>cryptic-name-1<delim>cryptic-name-2<delim>...
unit<delim>unit<delim>unit...
time<delim>data-1<delim>data-2...
.
.
.
```

The first line in the file is a tab or space delimited *header* containing the identifier '**t**' (for time) followed by the cryptic names for the independent variables.

The second line specifies the units that the data is expressed in (columns are separated by delimiters).

Subsequent lines must contain the time stamp, in decimal form and the data values (columns are separated by delimiters).

An example of a data file for the variables *bodcon* and *tkncon* for the stream with a label of *raw* is shown below.

```
t bodconraw tknconraw
d mg/L mgN/L
0.1 250 40
0.2 260 38
0.3 ? 37
0.4 275 ?
0.5 245 35
0.6 200 30
0.7 224 45
10.8 236 42
10.9 190 50
```

Figure 6-14 – Example of ASCII Data File

Excel Data Files

Data files can also be prepared using Excel. The format is similar to the ASCII text files, except that cells are used instead of delimiters to separate the data.

The first row in Excel is the header line that contains the cryptic names for the variables. Time ('t') is the first column.

The second row specifies the units that the data is expressed in.

Subsequent rows must contain the time stamp, in decimal form and the data values.

An example of a data file for the variables *bodcon* and *tkncon* for the stream with a label of *raw* is shown below.

	A	B	C
1	t	bodconraw	tknconraw
2	d	mg/L	mgN/L
3	0.1	250	40
4	0.2	260	38
5	0.3	?	37
6	0.4	275	?
7	0.5	245	35
8	0.6	200	30
9	0.7	224	45
10	10.8	236	42
11	10.9	190	50

Figure 6-15 – Example of an Excel Data File

New Data File Creation Tool

A tool has been included in GPS-X to simplify manual data entry in Excel by creating an Excel template file where the first two rows of data are populated by GPS-X.

To access the New Data File Creation tool:

(1) **Open the Data File Menu**. On the main toolbar press the Data File button to open the Data Files window.

(2) **Create a new Data File**. Pressing the New button on the Data Files window will open the New Data File Creation window. This will allow you to create a new Excel template file to be applied to the current scenario (see **Using Scenarios**) The New Data File Creation Window can be seen in **Figure 6-16**.

(3) **Choose the Type of Data File**. There are two types of data file variable types available:

- If the Input Variable option is selected, the variables available to add to the data file will be limited to the variables that currently have an input controller. The data entered in this file will change the value of the input controllers during the simulation.
- If the Output Variable option is selected, the variables available to add to the data file will be limited to the variables that are currently on an output display. The data entered in this file will be displayed

on the output graph to compare the simulation results to observed results (see **CHAPTER 8**).

Press **Next**.

(4) **Select the Variables**. All of the currently available variables will be grouped by the tab they are currently placed on in the options pane. Expand the tab name lists to select variables. When a variable is selected it will appear in the Current Selection pane.

When all variable selections are made, press **Next**.

(5) **Save the File**. By default, GPS-X will save the Excel file with the same name as the layout in the same directory. These defaults can be changed by pressing the **Browse** button.

Press the **Finish** button to save the file. You will be prompted to open the template file after it has been saved.

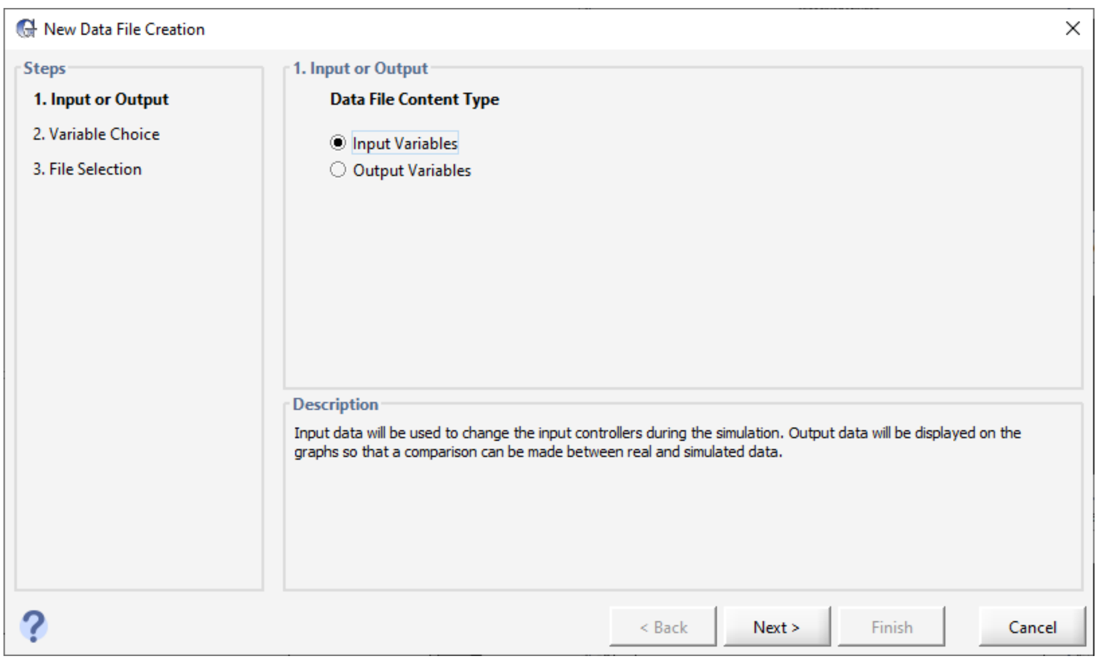


Figure 6-16 – Using the New Data File Creation Tool

Special Characters

? Fields in a data file that contain a question mark are interpreted as missing data. In this case, the last valid data input for that variable is used.

std If you enter a timestamp with the value 'std' that is interpreted as the value(s) to use at the start of a steady state simulation run. As opposed to the value '0.0' which is used at time zero (ie. the end of a steady state run – see **Figure 7-23** for an example)

Using the Data File Editor

GPS-X contains a simple tool for writing and editing *.dat* files (ie. ASCII text files with an extension of .dat). This tool can only create/edit a file that will be used for a single variable, so it's use is only suggested in simple cases or for beginners to GPS-X. More advanced users typically prefer to edit their files externally to GPS-X so that they can include many variables into one file.

The editor can be accessed in several different ways:

(1) **Through the Input Control Properties dialog.** When you change the desired controller to a file input type, a button will appear beside the Type box that will give you access to the editor.

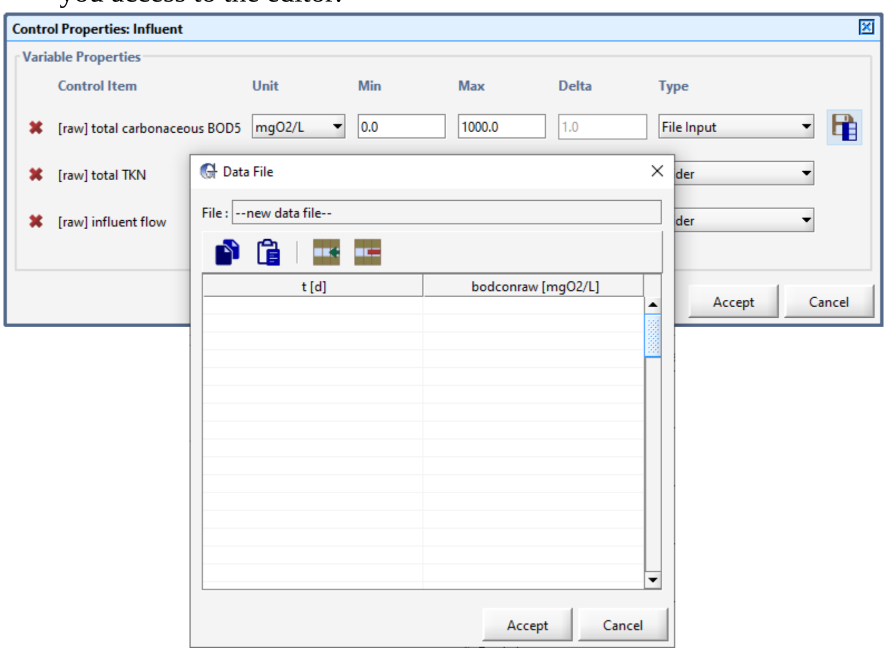


Figure 6-17 – Accessing the Editor through the Control Properties dialog

(2) **Right-clicking on the controller label.** This will pop up a menu where "Data File…" is one of the options.

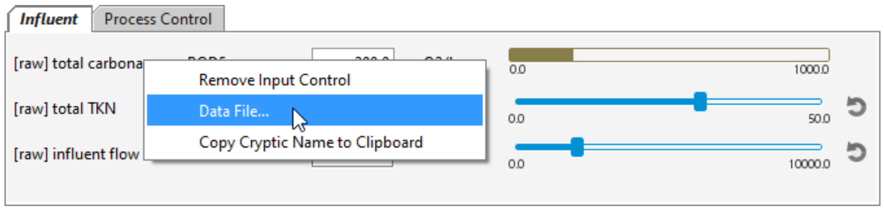


Figure 6-18 – Accessing the Editor through the Pop Up Menu

(3) **Right-clicking on the label in the data entry form.** This will pop up a menu where "Data File…" is one of the option.

ADDING INPUT FILES TO A LAYOUT

Once the data files have been set up properly you have to let GPS-X know that it should use these files during the simulation.

If you created the file using the GPS-X Data File Editor or the New Data File Creation tool, it is automatically added to the layout.

However, if you manually created the files external to GPS-X, you need to explicitly let GPS-X know that they exist. To do that:

(1) Access the "Scenario > Configuration" dialog window from the Simulation Toolbar (see **Scenario Configuration** in CHAPTER 8).

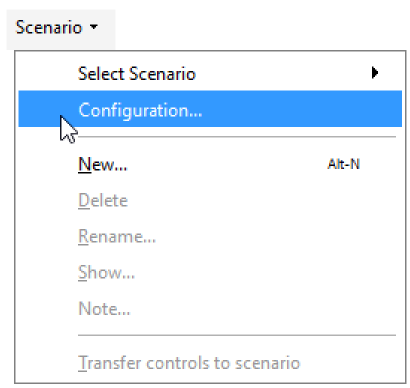


Figure 6-19 – Add Files through Scenario Configuration

(2) From the list of scenarios (see **Using Scenarios**), choose the appropriate one and click the "Data Files" button. This will display a dialog window with a listing of all the files that are currently being used by this scenario.

(3) From this dialog, you can choose to **Add**, **Remove**, or **Edit** a data file.

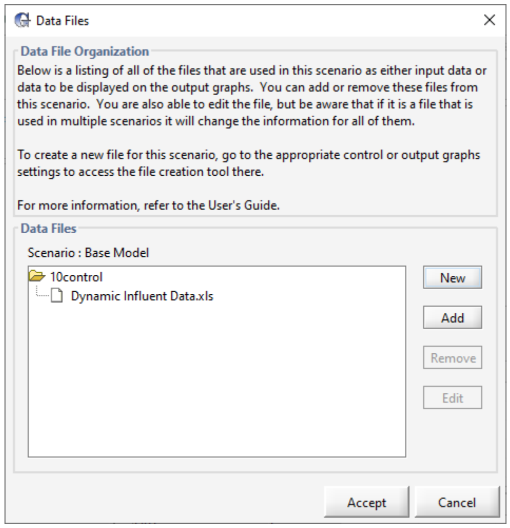


Figure 6-20 – Add, Remove or Edit Data Files

DYNAMIC DATA VALIDATION

When manually creating a dynamic influent characterization data set, it can be time consuming to verify each set of influent conditions results in a valid influent characterization. To simplify this, GPS-X is equipped with a tool to automatically verify that each set of influent conditions creates a valid influent characterization.

To use the Dynamic Data Validation tool:

(1) Open the influent advisor by right clicking on the Wastewater Influent object and navigating to *Composition* > *Influent Characterization*.

(2) Open the Dynamic Data Validation tool by pressing the Dynamic Data Validation button in the bottom left corner of the influent advisor.

(3) If you have manually prepared an influent data file, provide it to GPS-X by pressing the Browse button next to the Input File entry field. If you have not yet created an influent data file, press the Create Template button to have GPS-X provide you with a list of all the variables in the influent advisor. Select the variables that you have data for, and press accept. Save the file and populate it with data.

(4) Create an Output File. Pressing the Browse button next to the Output File data entry field will prompt you to save a new file that GPS-X has created. This file is where GPS-X will write the results of the Dynamic Data Validation.

The Dynamic Data Validation tool will apply the influent conditions described in your input file to the influent advisor at each time step. The values calculated by the influent advisor at each time step will be written to the output file. You will be prompted to open the output file once GPS-X has completed the Dynamic Data Validation.

In the output file any conditions that result in a influent advisor variable taking a negative value will result in those conditions being highlighted yellow. GPS-X will also highlight the negative values in that row red.

CHAPTER 7

Preparing Output

WHAT IS AN OUTPUT DISPLAY?

Output refers to the graphical or textual information generated by GPS-X. This can exist as a display or as a data file. Displays can be *static* or they can be linked to the simulator such that the display is *continuously updated* as the simulation progresses.

Output can also be saved to data files so that you can obtain prepare reports, store data in archives, etc. The sections below describe the types of output available and procedures for setting up and generating the output.

In **Simulation Mode**, the main window is divided up into three different regions; Controls, Outputs, and the Drawing Board (**Figure 7-1**).

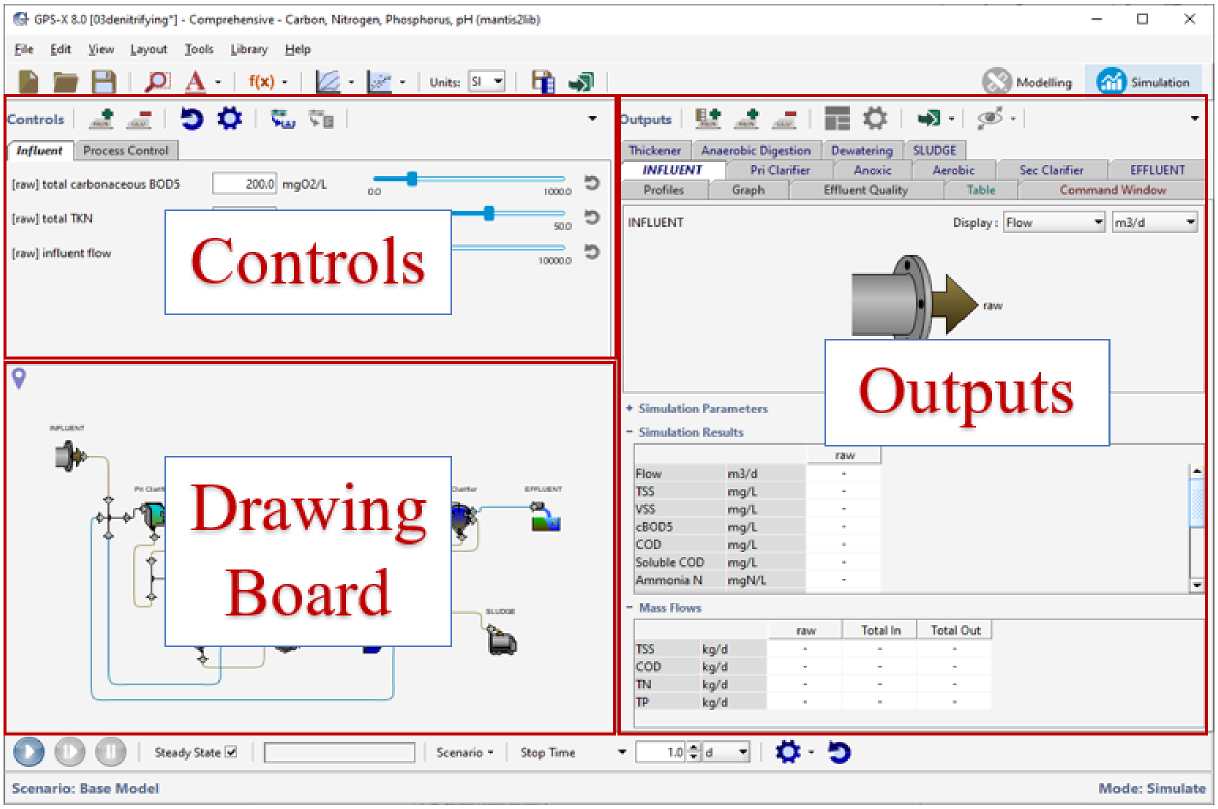


Figure 7-1 – Three Regions of Simulation Mode (Controls, Outputs, Drawing Board)

Output displays are set up and used in the **Simulation Mode**. They are organized into tabs. You can create as many tabs as you like and organize them in whatever way is convenient.

OUTPUTS TOOLBAR

At the top of the Outputs section of the main window is the Outputs toolbar. This gives you access to many features pertaining exclusively to the output displays.

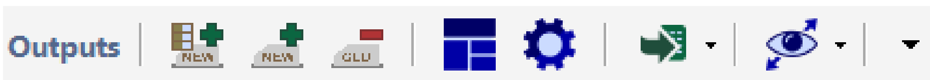


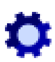
Figure 7-2 – Outputs Toolbar

New Table Tab –This will start a wizard to help you set up a table with your desired output variables. It will place the new table onto its own tab. See **Table Displays** for more information.

New Graph Tab –This will create a new tab for **User-Defined Displays**.

Delete Tab – This will delete the current tab and its contents.

Auto Arrange – This will automatically size and arrange the user-defined output displays so that they are all visible on their tab and cover the available space.



Output Properties… – See the **Output Properties** section below.

Export – This displays a drop-down list of the options available to export data from GPS-X for use in a report or spreadsheet.

Additional Output Displays – This displays a drop-down list of all available process schematic outputs. See **Process Schematic Output Summary**.

Tab Listing – This displays a drop-down list of all of the output tabs so that you can easily select the desired one. This can be useful when you have many tabs.

TYPE SUMMARY

There are several different types of output displays. Each output tab will contain one type of output and the label on the tabs have a different appearance depending on the type of output that it contains.

Quick Display

These panels provide a quick summary of the most important information for each unit process object in the model. See the **Quick Display** section for more information.

Table Displays

The tables are user-defined summaries of model outputs across the entire length of the plant (which can also be displayed in graphical form). See the **Table Displays** section for more information.

Bar Charts from Table Display

A visual representation of a row in the Table Displays can be created. See **the Bar Charts from Table Display** section for more information.

User-Defined Displays

The tabs that contain user-defined output displays can contain one or more graphs constructed from any output variable in the model. See the **User-Defined Displays** section for more information.

The following eight types of output displays are available to plot any output variable from any unit process:

1. ***X-Y*** *time series plot* – with time on the X-axis and the dependent variable on the Y-axis.
2. ***X-Y Scrolling*** *time series plot* – with the same axis labels as an **X-Y** time series plot and scrolling to the right with each time increment.
3. ***Bar Chart*** (primarily for array variables) – with each vertical bar representing a single array element's value.
4. ***Bar Chart (Horizontal)*** – similar to **Bar Chart**, but with the bars running horizontally, instead of vertically. Primarily for display of clarifier data.
5. ***Digital*** – displays only the current value of a variable.
6. ***3-D Bar Chart*** – displaying 2-D arrays where the z-axis is the array element's value.
7. ***Grayscale*** – where an element's value is associated with a shade of gray.
8. ***Probabilistic (Monte Carlo)*** – histogram representing percent probability of each bin.

In **Analyze Mode**, GPS-X substitutes the specified independent variable for time on all the time series displays to create a special X-Y plot. Refer to the **Creating User-Defined** section below, and CHAPTER 9 for more information on this type of graphic.

State Point Analysis

The secondary clarifier objects (circular and rectangular) provide options to plot **State Point Analysis Graphs**.

Send Data Directly to File

This option sends the simulation data directly to a plain text file. You may wish to do this to store simulation data for processing externally to GPS-X. See the **Saving Data to Text File** section for more information.

PROCESS SCHEMATIC OUTPUT SUMMARY

In addition to the output displays that are set up in the Output section of the main window, there are four types of Process Schematic Outputs that are shown in their own display windows since they require more space to visualize. The four types are:

Sankey Diagram

Five commonly used variables (Flow, TSS, COD, TN, and TP) can be displayed on a Sankey diagram. Sankey diagrams are flow diagrams that display variable quantities in terms of arrow width. This allows users to look at the plant's performance visually. See the **Sankey Diagram** section for more information.

Energy Usage Summary

This feature generates an energy usage summary based on the operational conditions and costs specified by the user. The values are displayed as a varying intensity of color around the unit process on the drawing board.

Operating Cost Summary

This feature generates an operating cost summary based on the operational conditions and costs specified by the user. The values are displayed as a varying intensity of color around the unit process on the drawing board

Mass Balance Diagram

Five commonly used variables (Flow, TSS, COD, TN, and TP) can be displayed on a Mass Balance diagram. Mass Balance diagrams are flow diagrams that display variable quantities in a tabular form. This allows users to look at the plant's performance numerically. See the **Mass Balance Diagram** section for more information.

QUICK DISPLAY

The **Quick Display** panels available in Simulation Mode are designed to present the important information to the user in the easiest manner possible. When you first build a layout, a Quick Display panel for every process on the drawing board is automatically created for you as a good starting point for setting up your outputs.

You can remove any of the Quick Displays that you aren't interested in by using the "Delete Tab" button on the Outputs Toolbar.

Double-clicking on any object in the drawing board will either create a Quick Display tab for that object (if it doesn't already exist) or select the existing Quick Display tab.

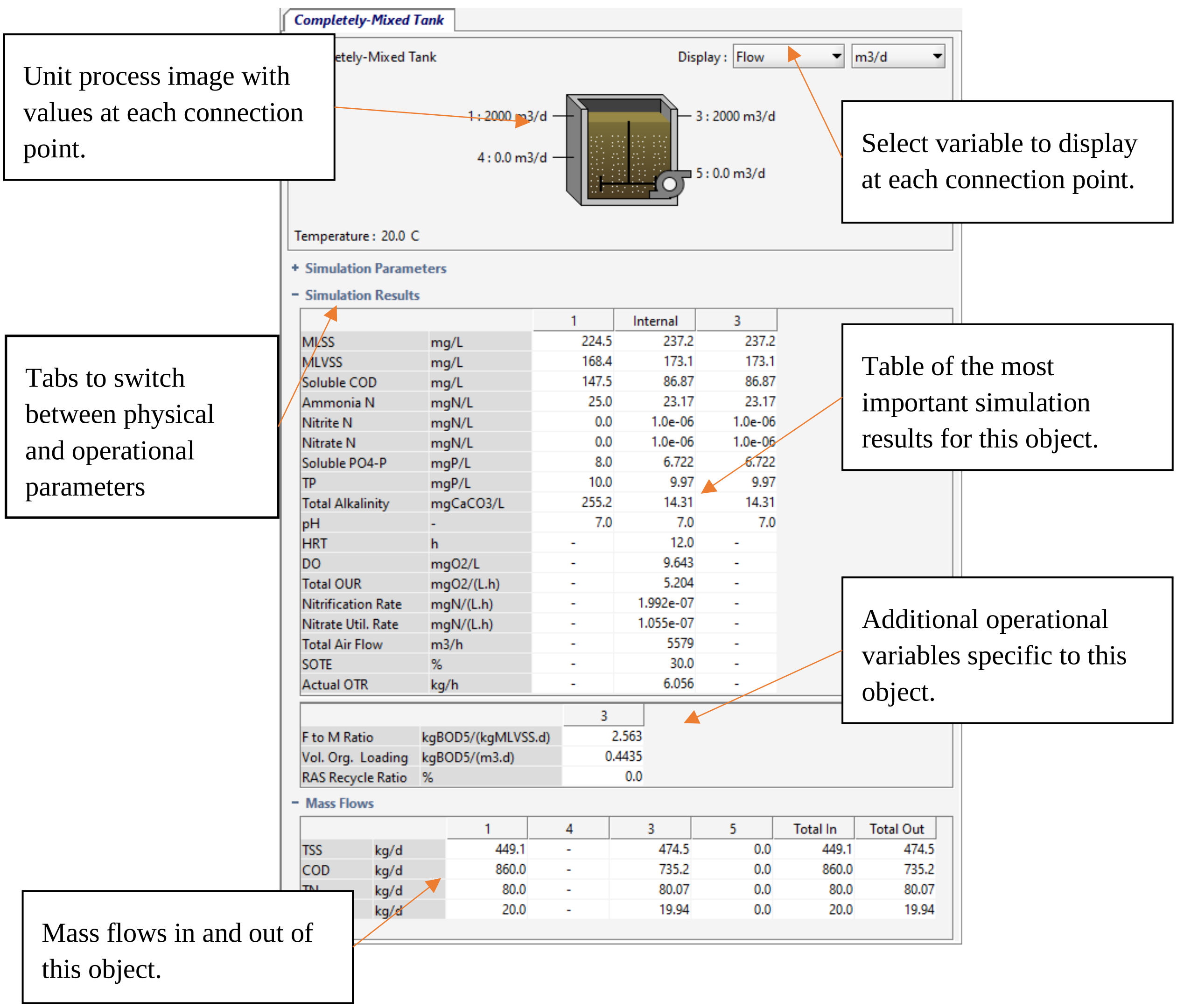
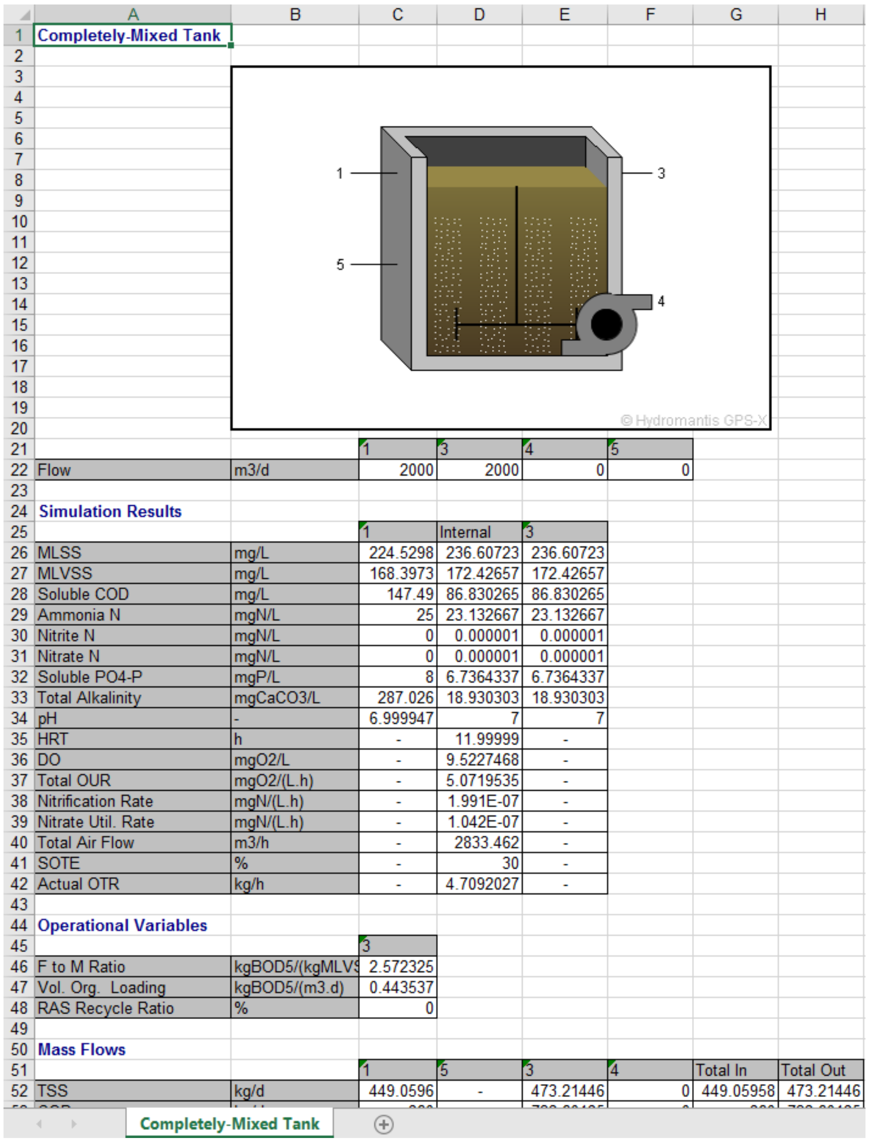


Figure 7-3 – Example of Quick Display

Exporting Data from Quick Displays

The entire Quick Display can be exported to an Excel spreadsheet by clicking on the "**Export Data to an Excel File"** menu item found in the **Export** button drop down list on the **Outputs Toolbar.** This will open a file browser where you can choose the name and location of the file.



A	B	C	D	E	F	G	H
Completely-Mixed Tank							
		1	3	4	5		
Flow	m3/d	2000	2000	0	0		
Simulation Results							
		1	Internal	3			
MLSS	mg/L	224.5298	236.60723	236.60723			
MLVSS	mg/L	168.3973	172.42657	172.42657			
Soluble COD	mg/L	147.49	86.830265	86.830265			
Ammonia N	mgN/L	25	23.132667	23.132667			
Nitrite N	mgN/L	0	0.000001	0.000001			
Nitrate N	mgN/L	0	0.000001	0.000001			
Soluble PO4-P	mgP/L	8	6.7364337	6.7364337			
Total Alkalinity	mgCaCO3/L	287.026	18.930303	18.930303			
pH	-	6.999947	7	7			
HRT	h	-	11.99999	-			
DO	mgO2/L	-	9.5227468	-			
Total OUR	mgO2/(L.h)	-	5.0719535	-			
Nitrification Rate	mgN/(L.h)	-	1.991E-07	-			
Nitrate Util. Rate	mgN/(L.h)	-	1.042E-07	-			
Total Air Flow	m3/h	-	2833.462	-			
SOTE	%	-	30	-			
Actual OTR	kg/h	-	4.7092027	-			
Operational Variables							
		3					
F to M Ratio	kgBOD5/(kgMLVS	2.572325					
Vol. Org. Loading	kgBOD5/(m3.d)	0.443537					
RAS Recycle Ratio	%	0					
Mass Flows							
		1	5	3	4	Total In	Total Out
TSS	kg/d	449.0596	-	473.21446	0	449.05958	473.21446

Figure 7-4 – Quick Display Exported to Excel

Additionally, the entire Quick Display can be exported to a Word document by clicking on the "**Export Tab to Word"** menu item found in the **Export** button drop down list on the **Outputs Toolbar.** This will open a file browser where you can choose the name and location of the file.

Alternatively, you can copy the data (text only) to the system clipboard by clicking the **"Copy Data to Clipboard"** menu item from the **Export** drop down list **Outputs Toolbar.** The data can then be pasted into a report or spreadsheet.

If you only wish to export a specific selection of the data, simply highlight the rows in the Quick Display table and then right-click on it. A popup menu with the options of either copying the selection or the complete table is presented.

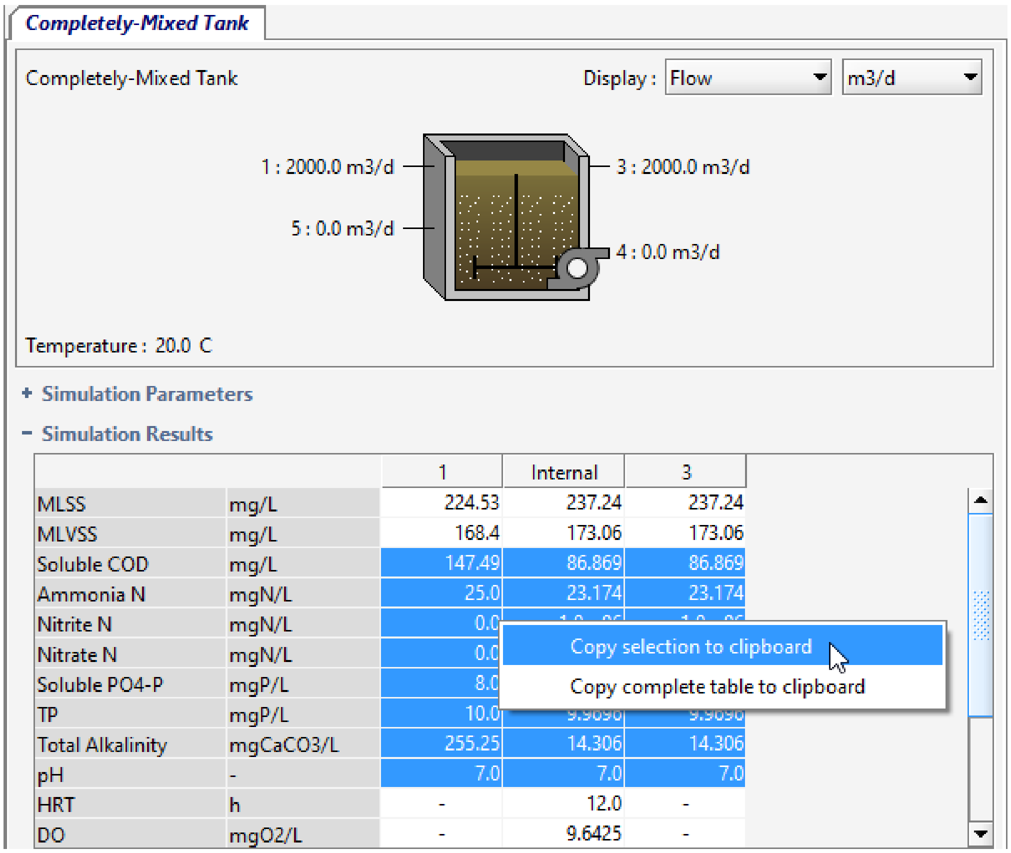


Figure 7-5 – Quick Display Copy to Clipboard

TABLE DISPLAYS

Reporting simulation data in a Tabular format is very convenient for preparing outputs for selected stream and process variables. The stream variable tables are very useful for viewing the mass balance of the most important plant variables across the whole plant.

There are two types of Table Displays:

Stream Variables

These tables display variables that are associated with the streams like flow rates, state variables and composite variables. An option is provided to tabulate the concentration values and/or the mass flows for the streams.

Process Variables

These tables display variables that are associated with the process itself like volume, HRT, organic loading, energy consumption, etc.

Setup Wizard

To create a Table Display, click on the **"New Table Tab"** button on the **Outputs Toolbar**. This will start the Table Properties Setup wizard (**Figure 7-6**).

(1) The first option to select under Table Type is whether you would like this table to display stream variables or process variables (see above for distinction).

(2) If this is going to be a stream variable table, you must decide whether to include the concentration values and/or the mass flows.

(3) You then simply select the location of the variables (left pane) and the specific variables themselves (right pane).

(4) When you are satisfied with your selection, click 'Accept' to create the Table Display tab (**Figure 7-7**).

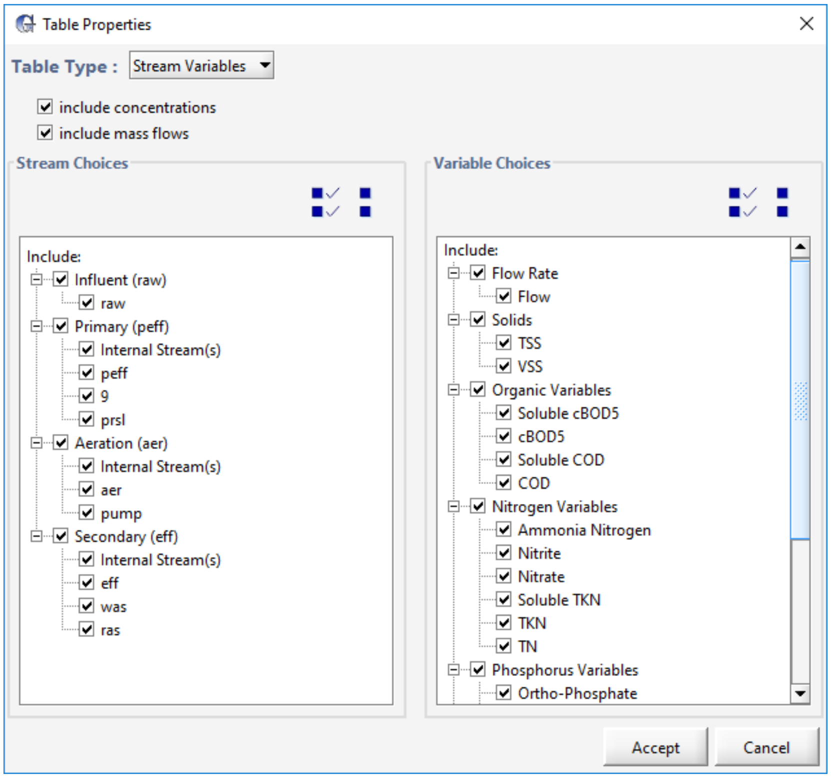


Figure 7-6 – Table Display Setup Wizard

Table: 1

		raw	peff	9	prsl	aer
Flow	m3/d	186100	185100	0.0	1000	385100
TSS	mg/L	234.0	83.32	0.0	30710	2605
	kg/d	43550	15430	0.0	30710	1003000
VSS	mg/L	175.5	62.44	0.0	23030	1878
	kg/d	32660	11560	0.0	23030	723200
Soluble cBOD5	mgO2/L	87.0	88.66	0.0	90.33	1.605
	kg/d	16190	16410	0.0	90.33	618.2
cBOD5	mgO2/L	213.6	134.2	0.0	16860	1088
	kg/d	39750	24850	0.0	16860	419100
Soluble COD	mgCOD/L	142.0	144.8	0.0	147.5	23.74
	kg/d	26440	26800	0.0	147.5	9142
COD	mgCOD/L	414.1	242.7	0.0	36190	2926
	kg/d	77080	44930	0.0	36190	1127000
Ammonia Nitrogen	mgN/L	24.55	25.82	0.0	25.0	0.2315
	kg/d	4568	4780	0.0	25.0	89.15

Figure 7-7 – Example of a Table Display

Exporting Data from Table Displays

The entire Table Display can be exported to an Excel spreadsheet by clicking on the "**Export Data to an Excel File"** menu item from the **Export** drop down list on the **Outputs Toolbar.** This will open a file browser where you can choose the name and location of the file.

Additionally, the entire Table Display can be exported to a Word document by clicking on the "**Export Tab to Word"** menu item found in the **Export** button drop down list on the **Outputs Toolbar.** This will open a file browser where you can choose the name and location of the file.

Alternatively, you can copy the data (text only) to the system clipboard by clicking the **"Copy Data to Clipboard"** menu item from the **Export** drop down list **Outputs Toolbar.** The data can then be pasted into a report or spreadsheet.

If you only wish to export a specific selection of the data, simply highlight the rows in the table and then right-click on it. A popup menu with the options of either copying the selection or the complete table is presented.

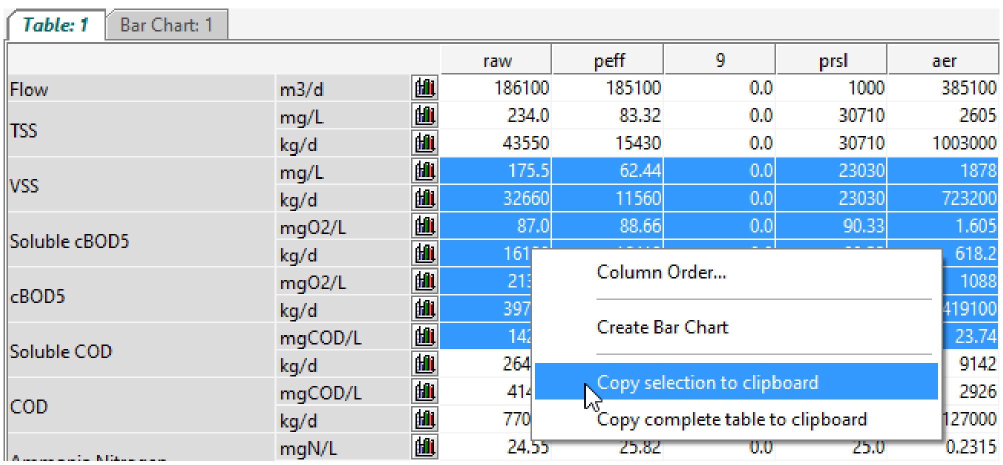


Figure 7-8 – Table Display Copy to Clipboard

Settings

Clicking on the "Output Properties" button on the **Outputs Toolbar**, will bring up the **Setup Wizard** where the current selections can be changed.

Column Order

Right-clicking on the Table Display will pop up a menu where one of the options is "Column Order…" (**Figure 7-8**). This will display a dialog window where you can change the default order of the column in the table.

BAR CHARTS FROM TABLE DISPLAY

You will notice that each row in the Table Display will have a small “**Graph**” button (**Figure 7-7**). This button creates a bar chart of that row’s data. A new tab will be created that shows all data in the selected row. (see **Figure 7-9**)

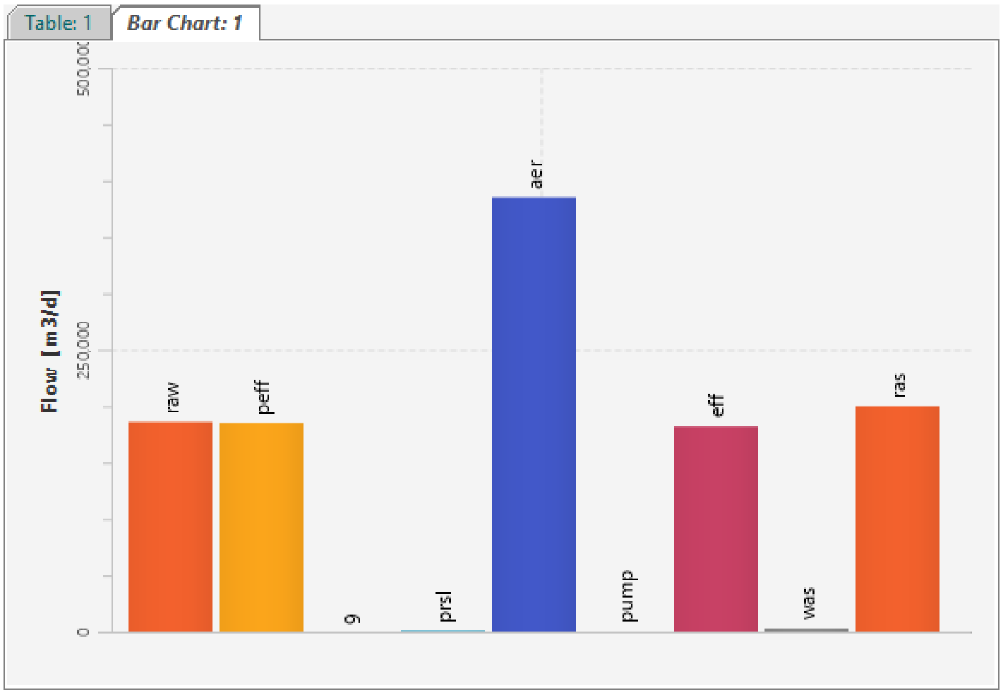


Figure 7-9 – Bar Chart created from Table Display

Settings

Clicking on the “Output Properties” button on the **Outputs Toolbar**, will display a dialog window where you can change various settings for the Bar Chart

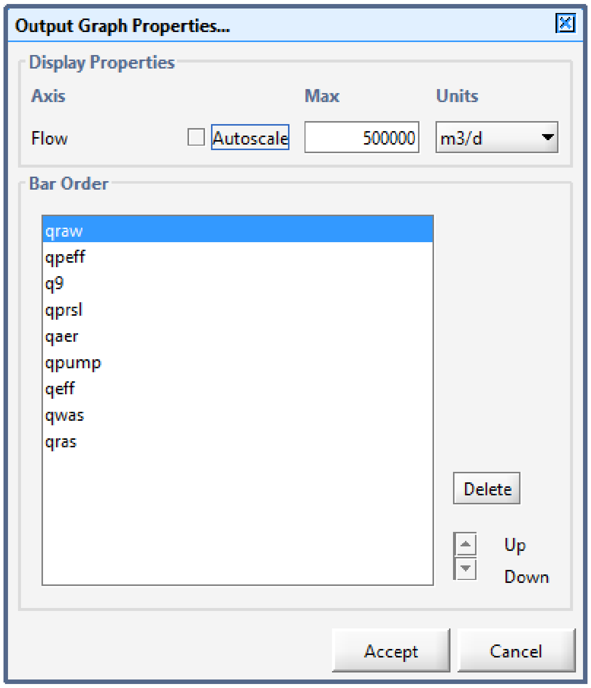


Figure 7-10 – Bar Chart Properties

The options include settings for the y-axis (autoscale or setting the maximum value, and units), as well as the order of the bars. You can also remove a variable from the chart by selecting it from the list and clicking "Delete".

USER-DEFINED DISPLAYS

You can create any number of user-defined graphs on multiple tabs. This allows users to plot additional information in formats that are the most useful for the simulation task at hand.

These displays must be created before running a simulation. This means that in setting up a simulation, you should consider the types of graphics desired, the variables to be plotted, and their minimum and maximum values. Think about the model behavior you would like to observe and ways to group output variables to maximize the information content.

Here are some guidelines for presenting data in output displays:

- ***Group Variables to be compared within a Single Graph***. For example, the state variable related to phosphorus removal in a single reactor can be compared easily if they are on the same display. Grouping encourages a visual comparison for the results.

- ***Display 6 or fewer Graphs on a Single Tab***. If you require a large amount of data to be plotted, create several tabs to allow fewer graphs per tab.

- ***Use Digital Displays when only the Instantaneous Value is Important***. For example, in some simulations, only the current solids retention time is important. Digital displays don’t provide information on rates of change but are convenient for displaying single values. Up to 20 variables can be displayed on a single digital display.

- ***Use X-Y or Scrolling X-Y Graphic Displays when the Past History, Level and Rate are Important***. These displays provide information on the past history (depending on the x-axis scale) of the variable, its instantaneous level and its rate of change.

- ***Use Bar Chart or Bar Chart (Horizontal) for Comparison of Levels and Rates of Change in Array Variables***. Many array variables are defined in GPS-X and most are so defined because of some spatial relationship between their elements, for example, the solids concentrations in each layer of a settler. Dynamic bar charts show the relative levels and rates of change for the variables in the array, but do not provide information on past history.

- ***Use Grayscale or 3D Bar Chart to Display DO profiles***, concentrations in a trickling filter, biofilm, etc.

- ***Use Problematic (Monte Carlo) graphs*** when displaying data from Monte Carlo simulations.

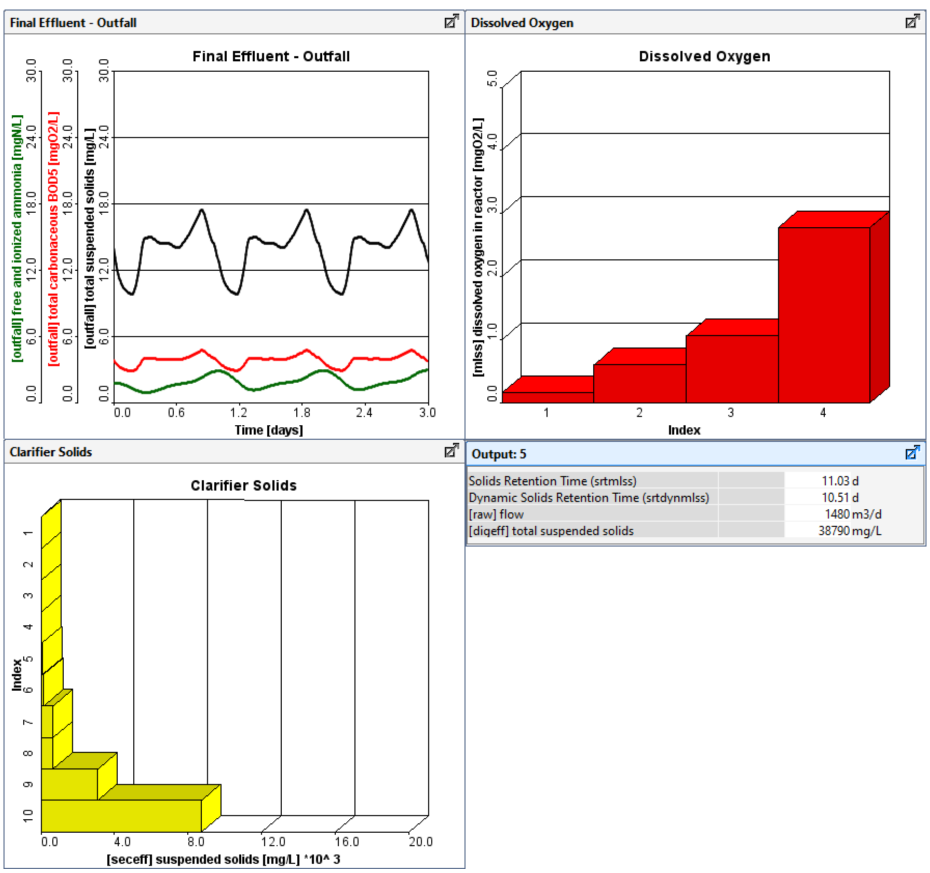


Figure 7-11 – Example of Different Types of Output Displays

It may take a few iterations to settle on an adequate display. You can stop the simulation at any time, right-click on output graphs to make changes, and then return to the simulation to view the new output. The specific procedures for these steps are examined in detail below.

Analyze/Optimize Graphs

The way an output graph is displayed depends on the simulation mode.

For example, in **Analyze Mode**, the graphics can be modified to accommodate data in a sensitivity analysis. In this case, the independent variable is not time, rather it is the variable defined by the user as the focus of the analysis. See CHAPTER 9 **Analysis Tools** for more information.

The display graphs also change in **Optimize Mode**. See CHAPTER 10 **Optimization Tools** for more information.

Creating User-Defined Output Displays

Once you have decided on the variables and kinds of graphics you would like to have displayed during a simulation, the next step is to instruct GPS-X to setup the output graphs. This step should be completed before running a simulation but you can iterate between the simulation and outputs setup to get the displays you like best. The outputs setup procedure is similar to that used for setting up input controls.

There are three major steps in setting up an output graph:

1. Create one or more blank output display tabs by clicking on the **"New Tab"** button on the **Outputs Toolbar**.
2. Locate the variables to be displayed on graphs, and drag them to the blank output tab.
3. Adjust the display properties.

Creating a New Output Display Tab

The **"Add Tab"** button on the **Outputs Toolbar** creates a new, blank tab. This blank tab can be filled with graphs containing variables from **Output Variables** menus from any unit process object in the layout.

Specify Output Variables to Display

Output variables include state variables and their derivatives, composite variables[7], inputs, special calculated variables, user-defined variables, and model constants.

Any of these variables, including system-related variables, can be displayed.

If the variable you need is not available, but can be calculated from existing variables, you can define it and make it available for plotting. For more information on defining your own variables, refer to CHAPTER 11 **Customizing GPS-X**.

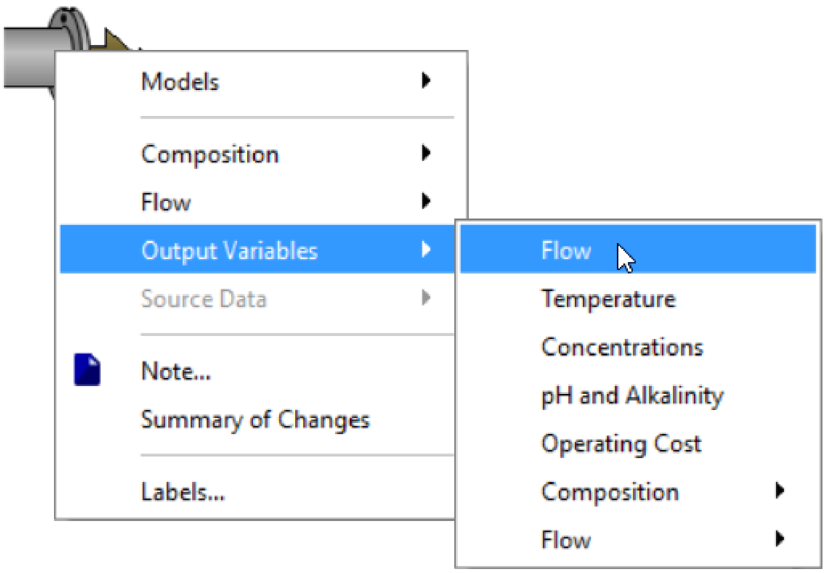


Figure 7-12 – Output Variables in Process Menu

To select a variable for display:

1. **In Simulation Mode,** bring up the **Process Data** menu by right-clicking on an object icon or one of its connection points. Different variables are available at the different spots. Note that the mouse cursor will change to an arrow when you are over top of a connection point.

[7] Composite variables are variables calculated from a combination of state variables and other constants.

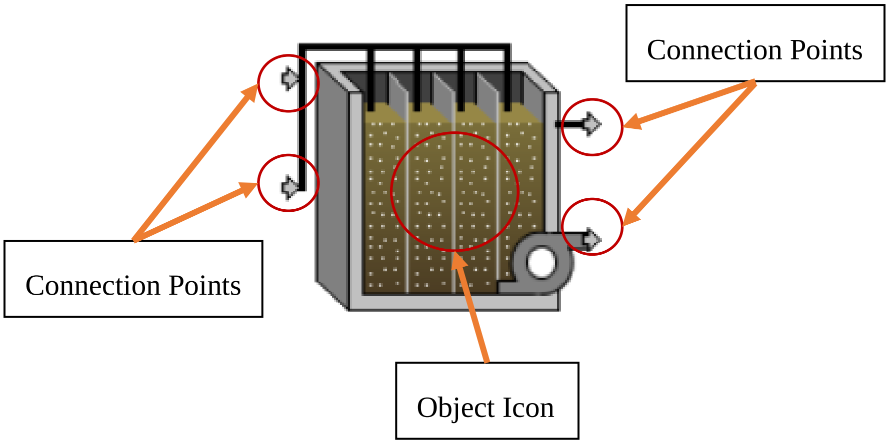


Figure 7-13 – Different Spots to Access Different Variables

2. **Select the *Output Variables* menu** as shown in **Figure 7-12**.

3. **Select the variable or variable group menu item of interest**. A dialog window will be displayed with a list of the available options.

   If the output variable is an array, as shown in **Figure 7-15**, click on the array button (**(…)**) to access a second dialog window for selecting the individual elements of the array.

4. **Find the display variable** and drag it to either

   i. an available blank area of the output tab (**Figure 7-14**) to create a new display. It produces an X-Y graph by default, but this can be changed at a later date (see **Output Properties** section).

   ii. or drop it on an existing display to add it to that one.

5. **Close the dialog** by clicking the ‘Accept’ button.

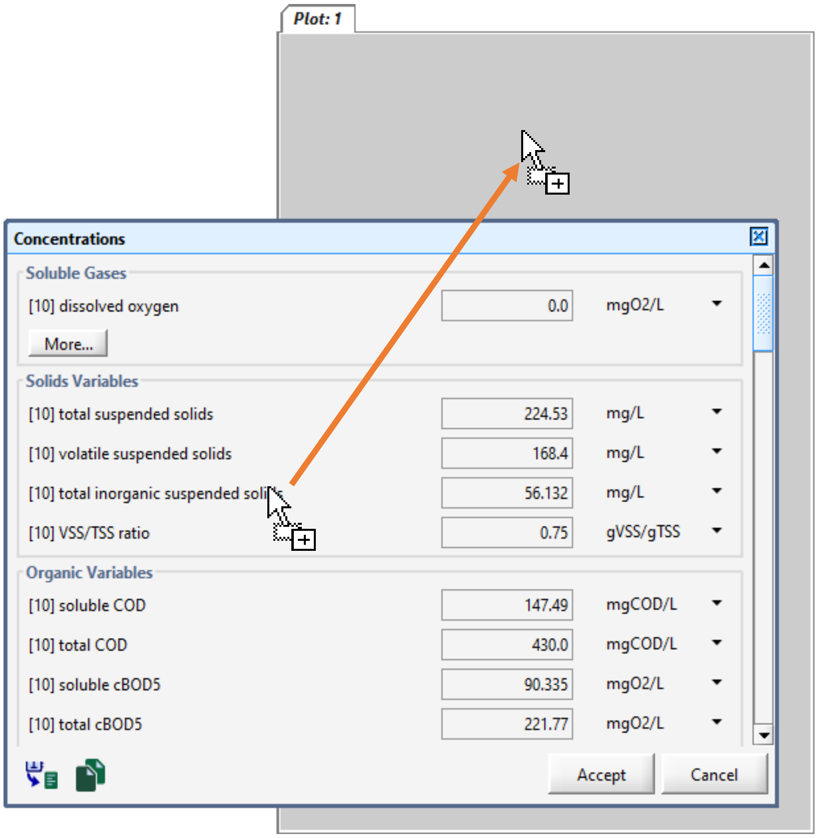


Figure 7-14 – Dragging Variable to Output Display

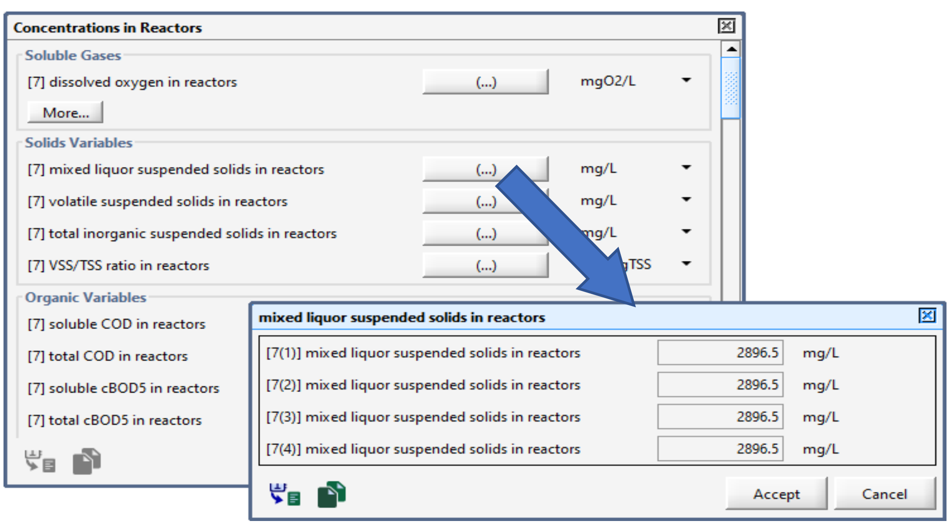


Figure 7-15 – Accessing Variable Array Dialog

Quick Tip: You can quickly create new output graphs by dragging a variable (from a menu or another input control window) to the area to the right of the output tabs. By doing so, a new tab will be created and the output variable assigned to it. This eliminates the need for clicking the **"New Tab"** button first.

Selecting Variables for Bar Charts

Elements of an array variable can be displayed most conveniently in a bar chart. To select an array variable for display, use the same procedure as above, but select drag the entire array (ie. the label beside the (…) button) to the output area instead of dragging the individual array elements.

Scalar variables can also be displayed in bar charts. In this case, the bar chart consists of a single bar; however, this is useful for displaying changes in sludge blanket heights, variable tank volumes, etc.

Exporting Data from User-Defined Output Displays

The entire Output Display can be exported to an Excel spreadsheet by clicking on the "**Export Data to an Excel File"** menu item found in the **Export** button drop down list on the **Outputs Toolbar.** This will open a file browser where you can choose the name and location of the file.

Additionally, the entire Table Display can be exported to a Word document by clicking on the "**Export Tab to Word"** menu item found in the **Export** button drop down list on the **Outputs Toolbar.** This will open a file browser where you can choose the name and location of the file.

Alternatively, you can copy the data (text only) to the system clipboard by clicking the **"Copy Data to Clipboard"** menu item found in the **Export** button drop down list on the **Outputs Toolbar.** The data can then be pasted into a report or spreadsheet.

Output Variables vs Input Parameters

The dialog windows that contain the output variables have a similar look to the data entry windows used to specify input values for a simulation. The output variable window differs in that simulation values are displayed, rather than showing an editable field for data entry. Keep this distinction in mind as you access the different window types to avoid confusing **output variables** with **input parameters**.

Display Variable Locations

Note the following conventions when selecting display variables:

- The pumped flow stream, effluent flow stream and last reactor in a plug-flow tank have the same composite, state and stoichiometric display variable values.

- Not all connection points have the same list of display variables, and the display menu inside the object itself is different from that of the connection points.

The display menus can be grouped into three broad categories:

1. Inside the object;
2. At the effluent or overflow connection point; and,
3. At all other connection points

These display menus are explained further in the next three sections.

Internal Display Menus

There are two types of internal display variable menus depending on the physical configuration of the object, as follows:

1. Completely mixed or zero-volume objects; and,
2. Objects with internal structure.

In the first type, there is no spatial variation in the objects.

The mathematical models are described by algebraic relations (for zero-volume objects) or ordinary differential equations (for completely mixed objects). In either case, no arrays are defined. The internal output variable menu is a copy of the menu at the effluent connection point.

Objects of the second type include plug-flow tanks, layered settlers, sequencing batch reactors, and others. Concentrations may have a gradient in these objects. In mathematical terms, partial differential equations are used and arrays are defined. In these objects, the internal output variable menu is used to access the one-dimensional arrays.

Effluent Display Menus

There are two types of effluent, or overflow, output variable menus depending on the physical configuration of the object, as follows:

1. Fixed-model objects (splitters, combiners, etc.); and,
2. All other objects

The display menus for effluent connection point of objects of the first type have the basic set of display variables, namely **Model Information, Flow, Composite Variables, State Variables** and **Model Stoichiometry**.

The overflow location is a special connection point for the majority of the objects. It has three sets of output variables:

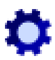
1. The basic display variables as for fixed-model objects;
2. Those internal model variables that are scalars, i.e., volume, sludge blanket, etc.; and,
3. Model parameters (just as in the Parameters menu, but only for display)

Other Display Menus

Display menus at these connection points have the basic set of display variables, namely **Model Information, Flow, Composite Variables, State Variables,** and **Model Stoichiometry**.

Output Properties

Once you have selected variables for display and placed them on one or more output displays, you can customize the properties and appearance of each graph.



This can be done by either clicking on the **"Output Properties…"** button on the **Outputs Toolbar** or right-clicking on an output graph to access the pop up menu and select the option from there.

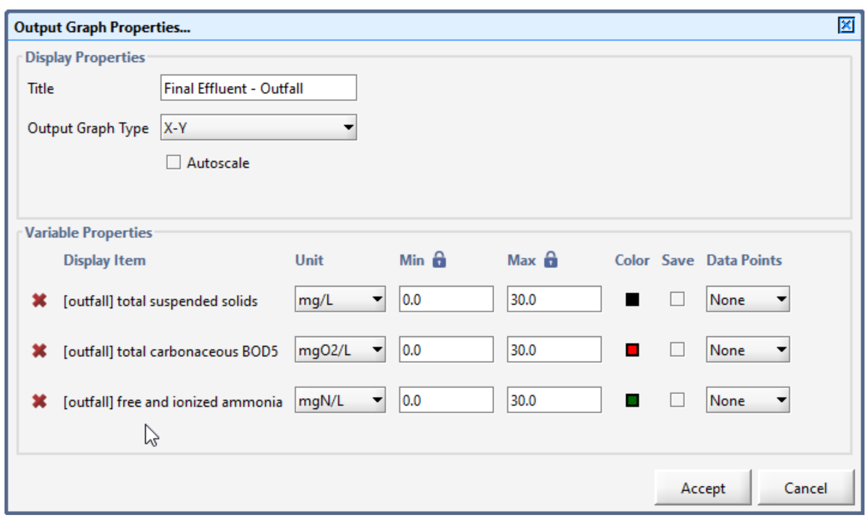


Figure 7-16 – Output Graph Properties Dialog

As shown in **Figure 7-16**, this dialog window has 2 sections.

Display Properties

This section contains settings for the whole graph, such as title, graph type and the autoscale option.

For a brief description of the different display types, see the **User-Defined Displays** overview section.

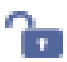
Autoscaling allows the program to determine the maximum value of the y-axis for you depending on the values of the variables during the simulation. This feature applies collectively to all of the y-axis in the graph and they are all given the same maximum value. The maximum value adjusts appropriately during the simulation run.

Variable Properties

The Variable Properties section contains individual settings for each of the variables in this display.

Note that some of the settings may be visible or not depending on the selections in the Display Properties section.

To **remove** the variable from the graph, click the small ‘x’ beside the variable you wish to remove.



If autoscale is not selected in the Display Properties section above, then you can explicitly state the minimum and maximum value that you’d like for the y-axis. Beside the min/max lable is a **lock icon**. If the column is ‘locked’ then that means that if you change one value, it will change it for the whole column. ‘Unlocking’ the column allows you to specify a different min/max for each variable.

The **Color** column allows you to specify the color of the lines or bars. Clicking on the small, colored box opens a color selector window that allows you to choose a different color.

The **Save** column allows you to automatically write data to a file located in the layout directory while the simulation is running for processing outside of GPS-X, such as preparing statistical summaries, presentation graphics or text documents. See the **Saving Data to Text File** section for more details.

The **Data Points** option allows you to display values from a file or database on the graph as points (as opposed to the simulation data which is displayed as lines). Selecting “File” will display a “Data File” button that can be used to create or view the data. The structure of the data file is the same as the files for File Input Controllers. See the **Manually Preparing Data Files** section for more information.

Quick Tip: The numerical value results from any graph can be instantaneously displayed by clicking on the graph itself.

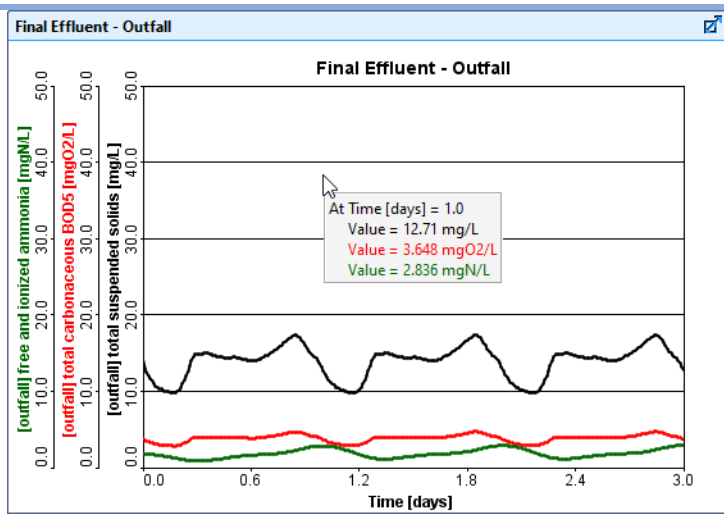


Figure 7-17 – Graph Showing Instantaneous Values

State Point Analysis Graphs

State point analysis is incorporated in the secondary settlers to analyze the maximum solid loading rates for design purposes. State point analysis may be conducted by using the simulated MLSS concentration or a user-defined design MLSS concentration.

To create a state point analysis graph, right click on the circular or rectangular clarifier objects, and select the ***Output Variables > State Point Analysis*** option, as shown below.

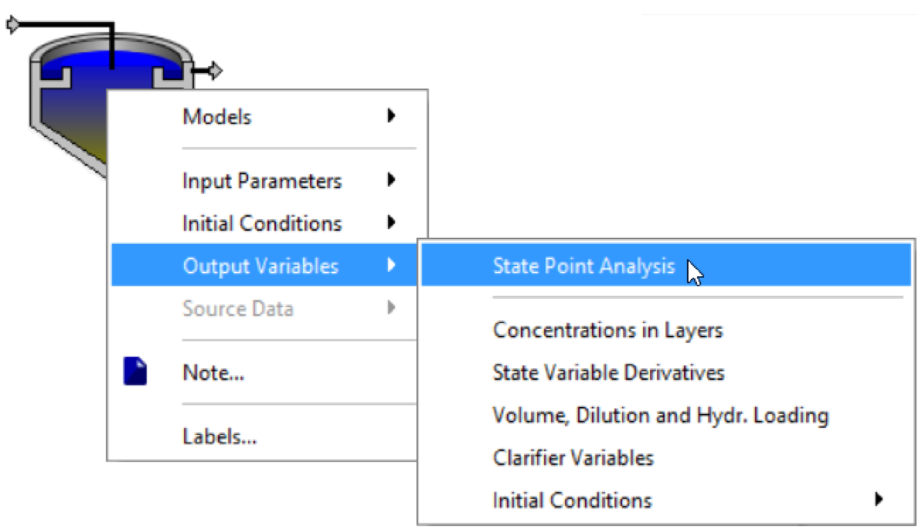


Figure 7-18 – Creating a State Point Analysis Graph

The state point analysis graph is dynamically updated for any fluctuation in the flow rates to the clarifier. A typical output of the state-point analysis is shown in **Figure 7-19**.

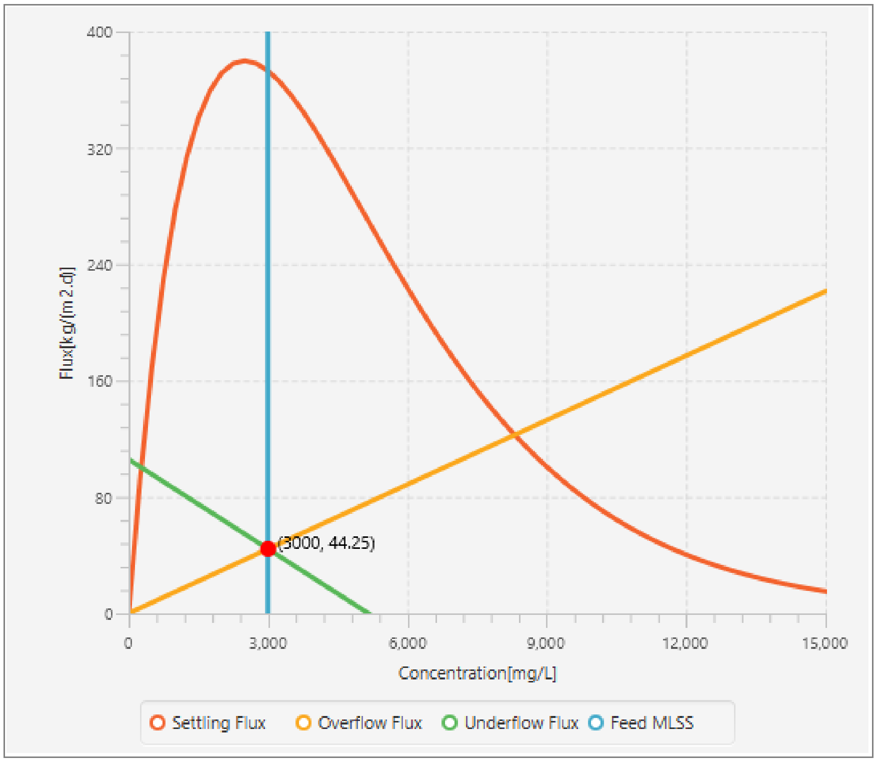


Figure 7-19 – Example of State Point Analysis Graph

As long as the operation point (red dot) is within the red curve, and the green line is not crossing the red curve, the operational condition of the clarifier is considered to be safe.

SANKEY DIAGRAM

After a simulation has been run, you can choose to view a Sankey diagram with the choice of five common variables (Flow, TSS, COD, TN, and TP).

Sankey diagrams are flow diagrams that display variable quantities in terms of arrow width. This allows users to look at the plant's performance visually and compare variable quantities.

Viewing a Sankey Diagram

To view a Sankey diagram, run the simulation and click on the **Sankey Diagram** menu item found in the **Additional Output Displays** button drop down list on the **Outputs Toolbar**.

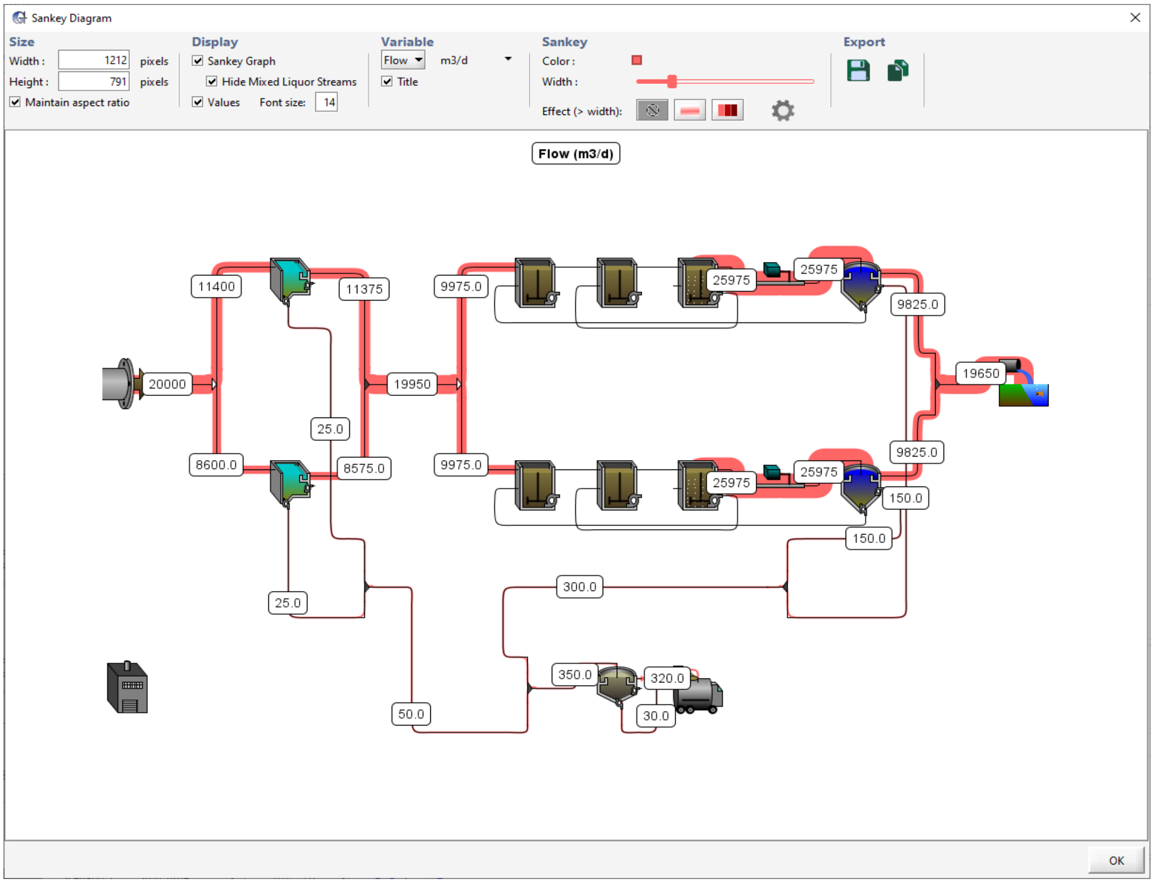


Figure 7-20 - Sankey Diagram Displaying Flow

Options for the Sankey Diagram

You can customize the look of the Sankey diagram in various ways and export the image to various formats. The options are grouped into five categories.

Size

In the size category, you can manipulate the size of the output diagram by defining the number of pixels you would like diagram width height to be. By selecting Maintain Aspect Ratio, GPS-X will automatically scale the unchanged dimension to maintain a constant ratio between the diagram width and height.

Display

In the display category, you can choose to hide/show the Sankey lines themselves and/or the numerical values that are displayed on the diagram.

If you are displaying the Sankey lines, you are also given the option to hide/show the mixed liquor streams. You may choose to hide these streams because the values of these streams are typically much larger than the other values and therefore overwhelm the visual difference in the other streams.

NOTE: You can also hide/show individual streams by moving the mouse cursor over the stream (the cursor will change to a hand) and right-click. If the stream is currently showing, you will be given the option to hide it and vice versa. Right-clicking on a blank area of the drawing board will give you to option to "Show All" currently hidden streams.

Variable

In the variable category, you can choose which variable to display and the unit that the value will be displayed in.

You can also choose to hide/show the title on the diagram where the variable name and unit are displayed.

Sankey

This category gives you various options on the appearance of the Sankey lines.

You can change the color by clicking on the little square that shows the current color.

You can vary the maximum width of the Sankey lines by adjusting the slider.

There are three effects that you can choose from for the Sankey lines.

1. **No effect**. This will just display the lines in the normal fashion.
2. **Fade**. A width limit can be set such that any value greater than that width will cause the line to fade from a solid color to a more transparent value.
3. **Darken**. A width limit can be set such that any value greater than that width will not increase the width of the line but will instead darken the color of the line proportional to the value.

> **NOTE:** The width limit for the fade/darken effect is set by clicking the settings button beside the effect option. This will pop up a dialog with a slider where you can adjust the limit and see the immediate effect on the diagram.

Export

This category gives you several options for exporting the image.

1. **Save Image to File.** Opens a file browser where you can choose the name and location of the image file to save the diagram to.

2. **Copy to Clipboard.** Copies the image to the system clipboard so that you can paste it into a report or spreadsheet.

MASS BALANCE DIAGRAM

After a simulation has been run, you can choose to view a Mass Balance diagram with the choice of five common variables (Flow, TSS, COD, TN, and TP).

Mass Balance diagrams are flow diagrams that display variable quantities in a tabular form. This allows users to look at the plant's performance numerically and compare variable quantities.

Viewing a Mass Balance Diagram

To view a Mass Balance diagram, run the simulation and click on the **Mass Balance Diagram** menu item found in the **Additional Output Displays** button drop down list on the **Outputs Toolbar**. This will start the Mass Balance Diagram wizard (**Figure 7-21**).

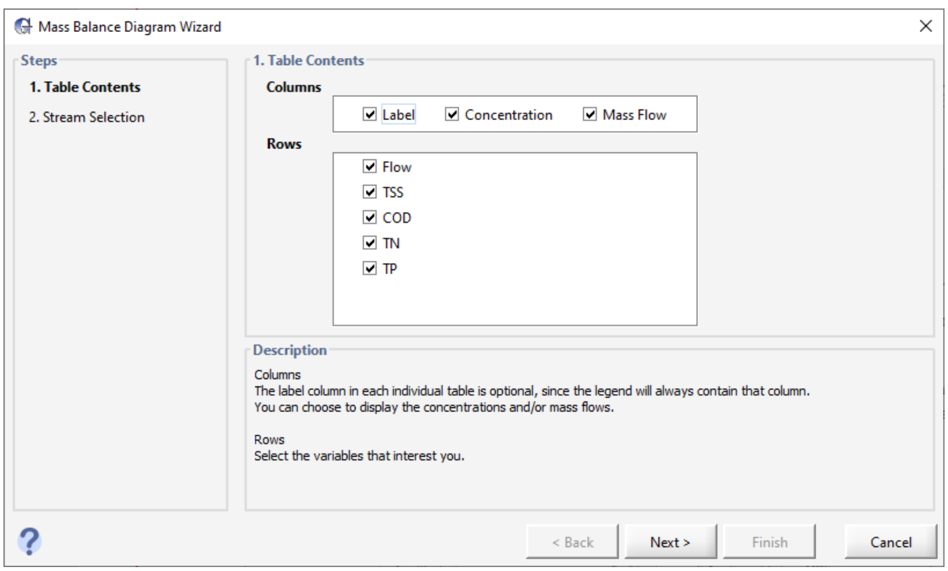


Figure 7-21 - Mass Balance Wizard Setup Menu

(1) The first option to select under Table Contents is what columns you would like to be displayed in each individual table (variable label, concentration and mass flow rate). You can then specify which variables you would like to be included in the tables (Flow, TSS, COD, TN, and TP) in the rows section.

(2) You then select the streams that interest you. When you are satisfied click Finish to create a Mass Balance Diagram.

(3) You will be asked if you would like to auto arrange the table locations, which will organizes the tables around the perimeter of the diagram close to the stream that they represent.

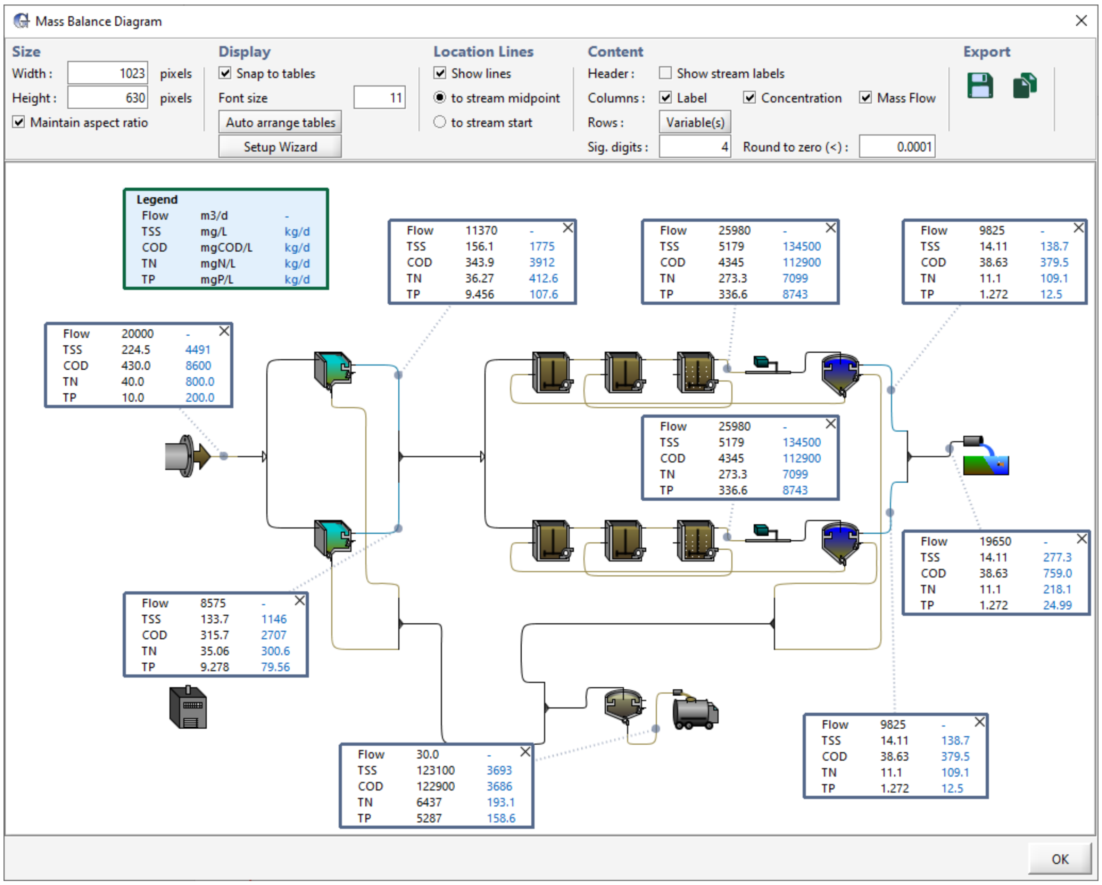


Figure 7-22 – Mass Balance Diagram

Options for the Mass Balance Diagram

Once the Mass Balance diagram has been generated, you can drag the tales anywhere you would like within the diagram.

You can further customize the look of the Mass Balance diagram in various ways and export the image to various formats. The options are grouped into five categories.

Size

In the size category, you can manipulate the size of the output diagram by defining the number of pixels you would like diagram width height to be. By selecting Maintain Aspect Ratio, GPS-X will automatically scale the unchanged dimension to maintain a constant ratio between the diagram width and height.

Display

In the Display category, you can choose to snap to tables which will attempt to align the tables with other tables or object in the diagram when you move them.

You can also choose to reopen the Setup Wizard in this category which will allow you to add or remove tables from the Mass Balance Diagram.

NOTE: You can also hide individual tables by moving the mouse cursor over the table and right-click. You will be given the option to hide the table.

Location Lines

In the Location Lines category, you can choose if you would like there to be lines connecting the tables to the stream they are representing.

You can also choose where you would like the location lines to connect (midpoint or start of the stream) if location lines are shown.

Content

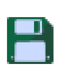
This category gives you various options for the content in the tables.

You can add the stream name as a header in each table by selecting the Showstream label option.

You can modify which column and row values selected in the setup wizard are shown in the tables.

You can also adjust the adjust the number the number of significant digits you would like present in the tables. Export

This category gives you several options for exporting the image.



3. **Save Image to File.** Opens a file browser where you can choose the name and location of the image file to save the diagram to.

4. **Copy to Clipboard.** Copies the image to the system clipboard so that you can paste it into a report or spreadsheet.

ENERGY USAGE AND OPERATING COST SUMMARY

After a simulation has been run, you can choose to view a plant schematic output summary of either the Energy Usage or Operating Costs. These two summaries are discussed together here since their appearance and settings are similar and the only difference is the type of data that is displayed.

Energy Usage Summary 

The model tracks power consumption for aeration, pumping, mixing, heating, and other.

To view an energy usage summary, run the simulation and click on the "Energy Usage Summary" menu item found in the **Additional Output Displays** button drop down list on the **Outputs Toolbar**.

Operating Cost Summary 

The model tracks operating costs for aeration, pumping, chemical dosage, sludge disposal, and miscellaneous.

To view an operating cost summary, run the simulation and click on the "Operating Cost Summary" menu item found in the **Additional Output Displays** button drop down list on the **Outputs Toolbar**.

Summary Views

The dialog window that is shown when choosing one of the options above, has two different views of the data.

Layout View

In layout view, an image of the layout is shown with 'hot spots' around the unit processes that represent the value of the variable that is being displayed. The intensity of the color of the 'hot spot' increases as the value gets larger.

An example is shown in **Figure 7-23** where the total power of each unit process is being displayed.

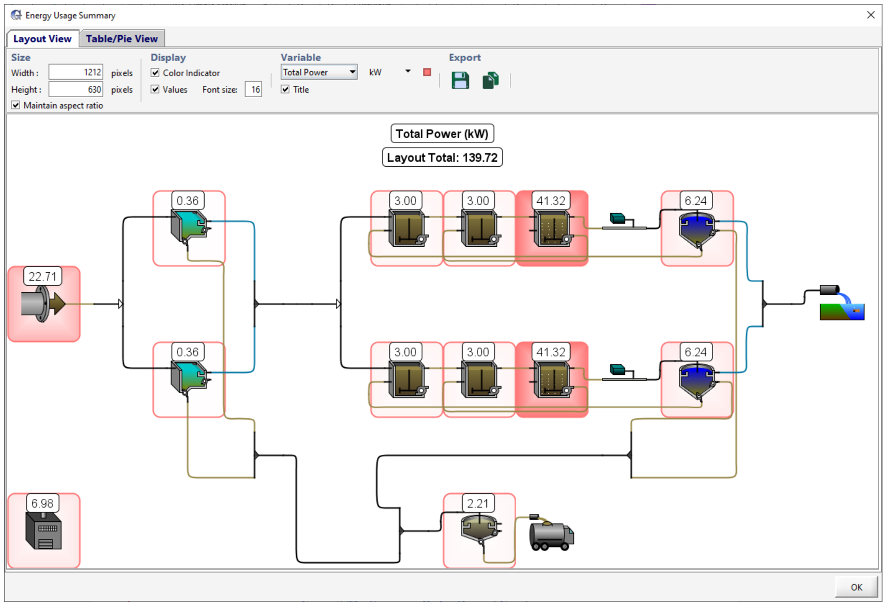


Figure 7-23 – Summary Dialog (Layout View)

Table/Pie View

In Table/Pie view, the window is divided in two with a table of the different values on the left and a pie chart representation of either the select row or column of data.

An example is shown in **Figure 7-24** where the row with the total power of each unit process is selected.

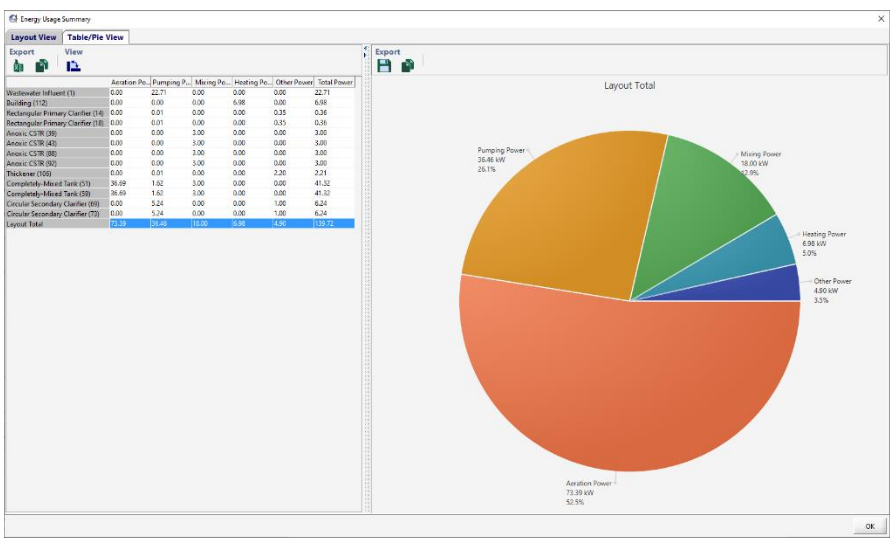


Figure 7-24 – Summary Dialog (Table/Pie View)

Options for the Layout View

You can customize the look of the plant schematic diagram in various ways and export the image to various formats. The options are grouped into four categories.

Size

In the size category, you can manipulate the size of the output diagram by defining the number of pixels you would like diagram width height to be. By selecting Maintain Aspect Ratio, GPS-X will automatically scale the unchanged dimension to maintain a constant ratio between the diagram width and height.

Display

In the display category, you can choose to hide/show the color indicator (ie. ‘hot spot’) and/or the numerical values that are displayed on the diagram.

Variable

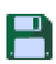
In the variable category, you can choose which variable to display and the unit that the value will be displayed in.

You can also choose to hide/show the title on the diagram where the variable name and unit are displayed.

An option to change the color of the ‘hot spot’ also exists. Clicking on the colored square box beside the variable option will display a color selection dialog.

Export

This category gives you several options for exporting the image.



1. **Save Image to File.** Opens a file browser where you can choose the name and location of the image file to save the diagram to.

2. **Copy to Clipboard.** Copies the image to the system clipboard so that you can paste it into a report or spreadsheet.

Options for the Table/Pie View

Export

Similar to above, you can export the data/image in various forms.

View

The only **View** option is the ability to change the highlighted selection between rows and columns. You can either use the button on the toolbar or you can click on the row or column header to switch it.

SAVING DATA TO TEXT FILE

In addition to the Microsoft Excel reports that can be generated for any graph, the simulation results can be saved directly to a plain text file (tab delimited) while a simulation is running.

In the **Output Properties** dialog window (**Figure 7-16**), there is an option to '**Save'** the variable. If this check box is selected, the data from the simulation will automatically be saved to a file.

All variables that are to be saved will be written to the same file that is located in the same directory as the layout file. The name of the file will have the following format:

`layoutname_scenarioname_yyyy_mm_dd.out`

Where `layoutname` is the name of the current layout, `scenarioname` is the name of the current scenario and `yyyy_mm_dd` is the date specified in the Site Properties window under the "Simulation Setup" tab.

Contents

The columns in the file are delimited by tabs.

The first line contains a list of the selected variables, starting with the identifier `'t'` to indicate the time column.

The second line lists the units of each variable.

If a steady-state evaluation was performed before the run begins, the third line begins with the identifier `'std'` followed by the results of the steady-state evaluation for each variable.

The remaining rows contain the time stamp and value(s) for each variable.

```
t xltank(1) xltank(2) xltank(3)
d g/m3 g/m3 g/m3
std         1849.7      1835.5      1822.8
0           1849.7      1835.5      1822.8
0.02        1849.7      1835.5      1822.8
0.04        1849.7      1835.4      1822.8
0.06        1849.6      1835.4      1822.8
0.08        1849.6      1835.4      1822.8
0.1         1849.6      1835.4      1822.8
0.12        1849.6      1835.4      1822.8
```

Figure 7-25 – Example of Plain Text Output File

The `.out` file remains open until you return to Modelling mode or close the layout.

If **Start** is selected more than once, any existing file will be overwritten; therefore, the `.out` file will always contain the results of a single simulation run from the starting time to the stopping time at the interval indicated by the value of the **Communication** interval.

If desired, you can rename the output data file or move it to a different directory so that subsequent simulations do not overwrite the file.

Using Output Data as Input Data

As GPS-X output data files have the same format as input data files, you can use the input/output capabilities to emulate a record/run macro.

To record a simulation session:

1. **Set-up the variables you want to record for file output.** In this case, these would be variables that you set up as interactive controls (eg. sliders).
2. **Run a simulation, changing the values of the controllers as you like**.
3. **Rename the output data file to conform to the input data file convention**. To do this, simply change the extension of the file from `'out'` to `'dat'`.
4. **For each variable specified in Step 1,** change the input controller from the interactive control to a File Input control and run the simulation.

When you run the simulation in step 4, the variable values will be read from the input data file, essentially reproducing the simulation you performed in step 2.

This technique is useful for generating artificial scenarios.

For example, if you want to generate data which approximate a storm event, you can use this method to save data as you manually change the influent flow. Later you can use this data for testing the effects of the storm flow.

GENERATING A REPORT

Reports can be exported to a Microsoft Excel spreadsheet file or a Microsoft Word document. These reports can contain images, data, parameter values, model results, and any other combination of text and images.

The report can be generated either before or after simulation.

Reports generated in **Modelling Mode** will contain only the information about the model and parameter defaults.

Reports generated in **Simulation Mode** can also contain information about the current state of the model, and time series data from the most recent simulation.

There are three types of reports:

1. Standard Word Report
2. Standard Spreadsheet Report
3. Custom Spreadsheet Report

You can access the report dialog by either selecting Report from the **File Menu** or clicking on the Report button on the **Main Toolbar.**

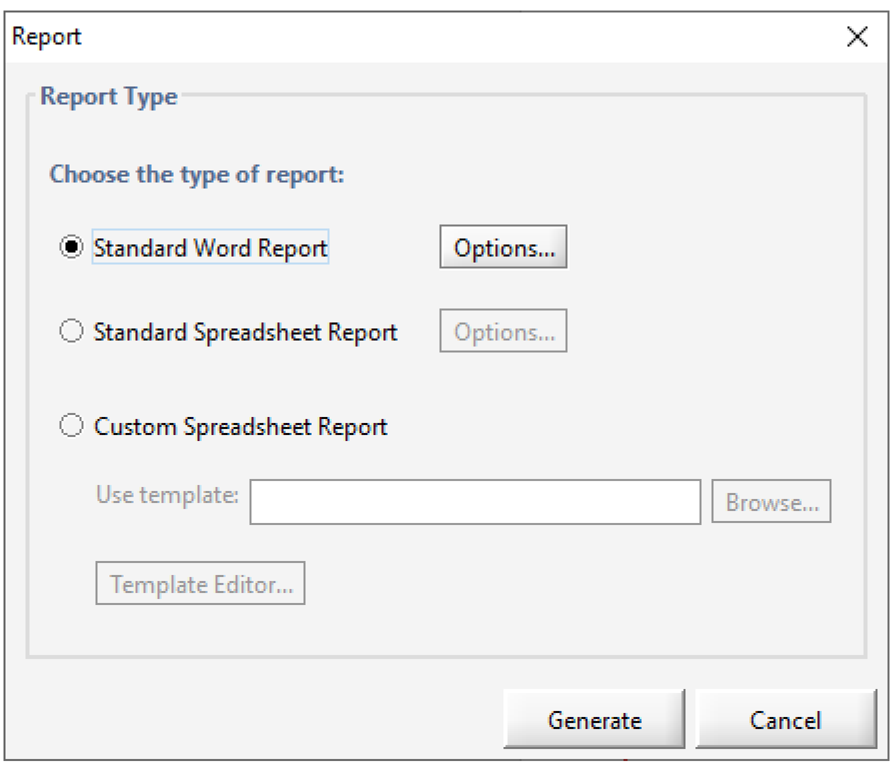


Figure 7-26 – Report Dialog

Standard Word Report

Standard Word reports use a fixed format and style to display all model input, output and simulation results.

By default, all information will be included in the report. If you wish to exclude certain types of information, select the **"Options…"** button.

The **Options Dialog** (**Figure 7-27**) allows users to specify which types of information are to be included in the standard Word report. The information is organized under multiple headings within the report.

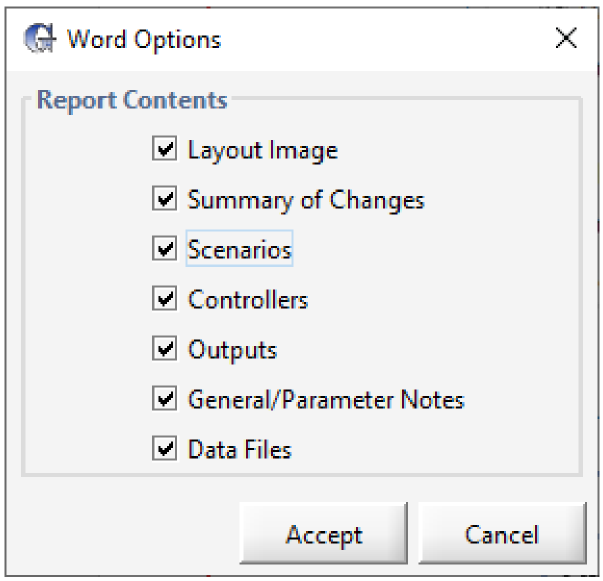


Figure 7-27 – Options Dialog (Standard Word Report)

> **Quick Tip:** In cases where a user has entered information into the **"Notes"** field for a parameter that information is inserted included in the General/Parameter Notes section of the report, where each object with a Note will be given a table.

Standard Spreadsheet Report

Standard spreadsheet reports use a fixed format and style to display all model input, output and simulation results.

By default, all information will be included in the report. If you wish to exclude certain types of information, select the **"Options…"** button.

The **Options Dialog** (**Figure 7-28**) allows users to specify which types of information are to be included in the standard spreadsheet report. The information is organized into multiple worksheets within an Excel file.

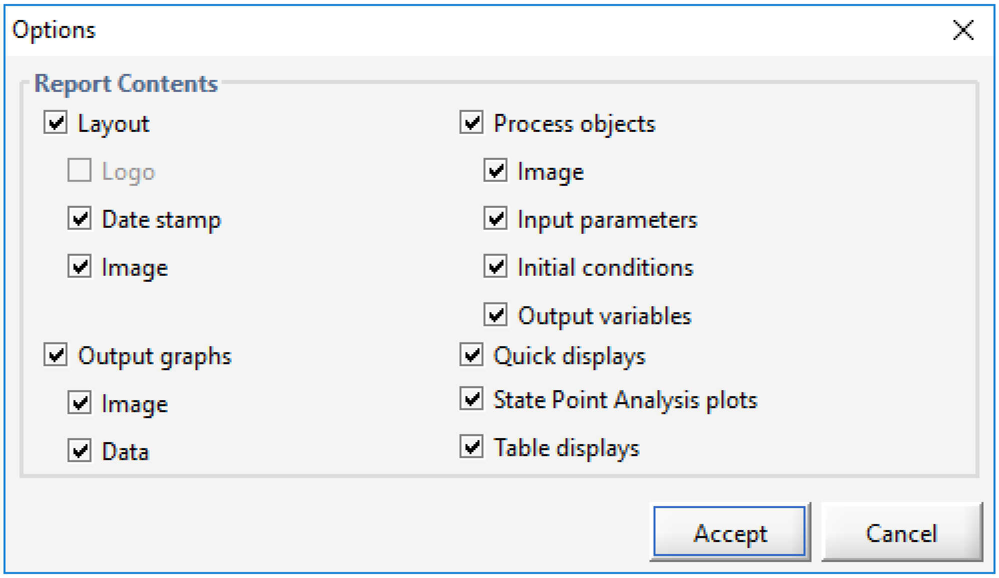


Figure 7-28 – Options Dialog (Report to Standard Spreadsheet)

> **Quick Tip:** In cases where a user has entered information into the **"Notes"** field for a parameter that information is inserted into the report as an Excel comment attached to the cell that contains the label of that parameter

If you would like a company logo automatically inserted on the front page of the report, place the image (in ***.bmp, .gif, .jpg, .png,*** or ***.tiff*** format) in the following sub-directory of the GPS-X installation directory:

```
\bin\gpsx\resources\images\report
```

Custom Spreadsheet Report

The custom report option uses an XML-type template to specify which data are inserted into the report, with its location and formatting. A good understanding of the XML format is assumed if you are going to use this custom report option.

The template file (with the extension "`.gpr`") contains a series of tags that describe how the report is to be laid out.

To generate a custom report, select the **Custom Report** option from the **Report Dialog** (**Figure 7-26**). You will be prompted to either browse to an existing template or to create a new template with the **Template Editor**.

Template Editor

Clicking the **Template Editor** button on the **Report Dialog** starts the template editor.

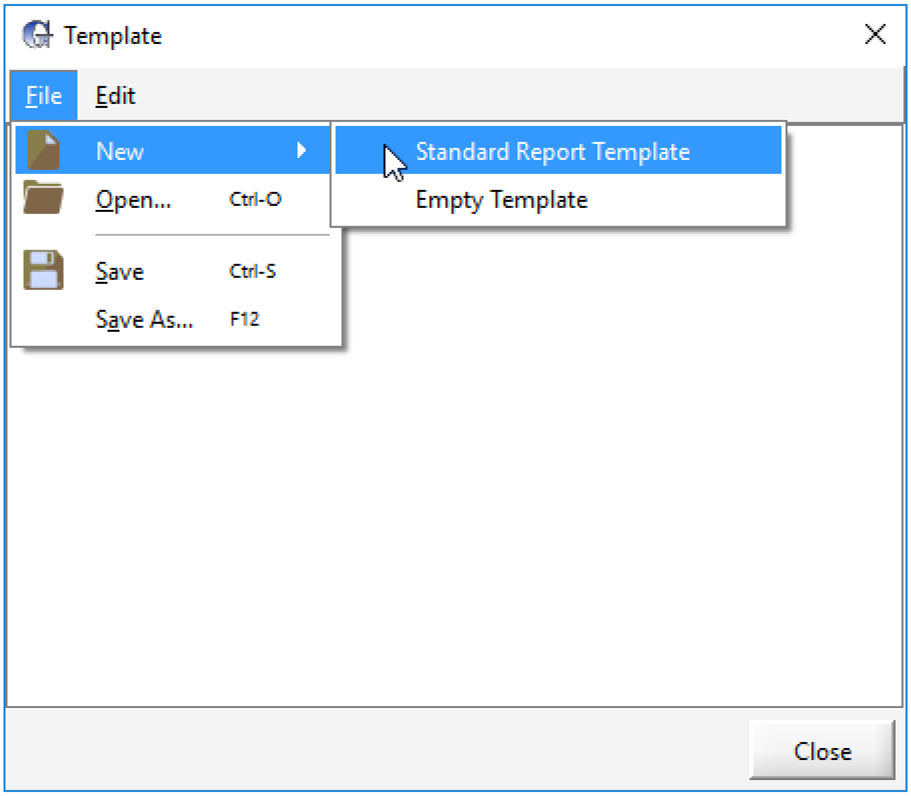


Figure 7-29 – Template Editor

In most cases, you'll want to start with the standard report template and then proceed to organize and add/remove items to customize it to your preferences.

To create a new template, select "New" from the File menu and choose "Standard Report Template" to populate the editor window with the XML specification of the currently selected report options.

Quick Tip: You can access the standard options through the ***Edit > Standard Template Options*** menu item to include/exclude certain items before creating a new template to begin from a closer starting point to your desired results.

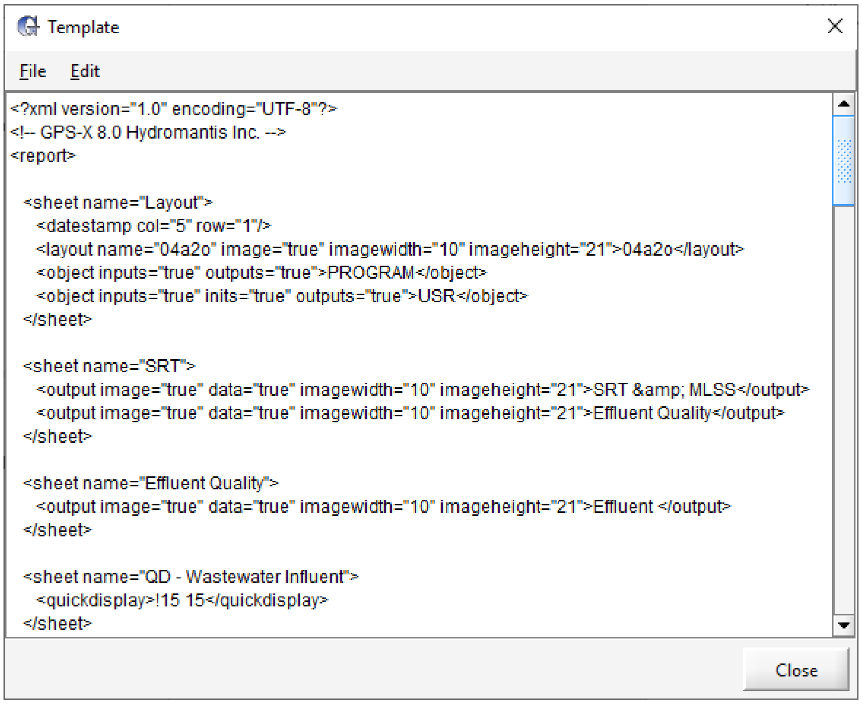


Figure 7-30 – Example of a Template File

The GPS-X report template will contain various XML tags.

The report template **<u>must</u>**:

- Start with a <xml version="1.0" encoding="UTF-8"?> header tag
- Include exactly one <report> </report> tag pair that encloses the entire customized XML report specification.

Each <report> contains one or more <sheet> </sheet> tag pairs.

A sheet tag corresponds to one Excel sheet. Each <sheet> tag can contain one or more of the following tags:

- <image>
- <datestamp>
- <layout>
- <object>
- <output>
- <data>
- <value>
- <text>

The following sections describe the use of the above tags:

XML	Tag	Description
<sheet name=””> </sheet>	**name**	Label of sheet tab
<image imagepath=”” imagewidth=”” imageheight=”” cell=”” />	**imagepath**	Path to location of image file (bmp, gif, jpg, png, tiff) i.e. C:\images\image.jpg
	imagewidth	Number of columns the image will span
	imageheight	Number of rows the image will span (if imagewidth and imageheight are not specified, the image will be inserted at the original aspect ratio)
	cell	Place top left corner of image in cell(col,row), i.e. cell=“A4”
<datestamp cell=””/>	**cell**	Insert date/time in cell(col,row) i.e. cell=“A4”
<layout name=”” cell=”” image=”true|false” > layoutname </layout>	**layoutname**	GPS-X layoutname
	name	Layout title. Defaults to “layoutname”
	image	Show content of GPS-X drawing board. Defaults “true”
	cell	Cell(row,col) i.e. cell=“A4”
<object image=”true|false” inputs=”true|false” inits=”true|false” outputs=”true|false” > objectid </object>	**objectid**	Is a process object ID (see Table 1 for valid IDs)
	image	Show process table image for process. Defaults “true”
	inputs	Show process input parameters. Defaults “true”
	Inits	Show process initial conditions. Defaults “true”
	Outputs	Show display variables. Defaults “true”

XML	Tag	Description
<output image="true|false" imagecell="" imagewidth="" imageheight="" data="true|false" cell="" stats="true|false" > graphname </output>	**image**	Show image of graph. Defaults "true"
	imagecell	Top left corner of graph image i.e. cell="B10"
	imagewidth	Number of columns image will span
	imageheight	Number of rows image will span
	data	Display data from graph. Defaults "true"
	cell	Top left corner of data column(col,row) i.e. cell="A4"
	stats	Display statistics (min, max, mean, std dev). Defaults "true"
	graphname	Title of graph
<data cell="" xdata="true|false" stats="true|false" npoints = ""> variables </data>	**cell**	Location of data to be inserted i.e. cell="A4"
	xdata	Inserts X-axis (e.g. t) data. Defaults "true"
	stats	Show statistics (min, max, mean, std dev). Defaults "true"
	npoints	Show only the last npoints number of data points in the time series. Default is to show the entire time series. To be inserted into the report, all variables must be on visible graphs
	variables	Space delimited list of cryptic variable names
<value name="" cell="" stream=""> variables</value>	**name**	Descriptive label
	cell	Insert current value, starting at (col, row) i.e. cell="A4"
	stream	Stream label associated with variables
	variables	Space delimited list of cryptic variable names.

XML	Tag	Description
<text value="" cell="" bold="true|false" italics="true|false" color="" bgcolor="" align="" size="" />	**value**	Text string to be inserted
	cell	Location to insert text i.e. cell="A4"
	bold	Bold text formatting
	italics	Italic text formatting
	color	Text color (default is black)
	bgcolor	Background cell color
	align	Alignment of text within the cell options are Left, Right, Centre Fill, General, Justify
	size	Size of font

Available color names are:

black	bright green	sky blue
white	sea green	pale blue
red	dark green	rose
dark red	olive green	lavender
yellow	teal	tan
dark yellow	dark teal	aqua
light blue	grey 25%	lime
pink	grey 40%	gold
turquoise	grey 50%	orange
light turquoise	grey 80%	light orange
green	blue grey	brown
light green	dark blue	indigo
blue	violet	plum

STATISTICAL ANALYSIS

If you have set it up so that measured data is being plotted on a graph (see Data Points in the **Variable Properties** section) then you can perform a number of statistical analyses of the difference between the simulation and the measured data.

These evaluations provide you with an estimate of the quality of the simulation (compared to measured data) during the process of model calibration.

Setting Up a Statistical Analysis

For any X-Y (time series) graph that contains data read from a file, a statistical analysis can be set up by right-clicking on the graph, and selecting the **Statistics** menu.

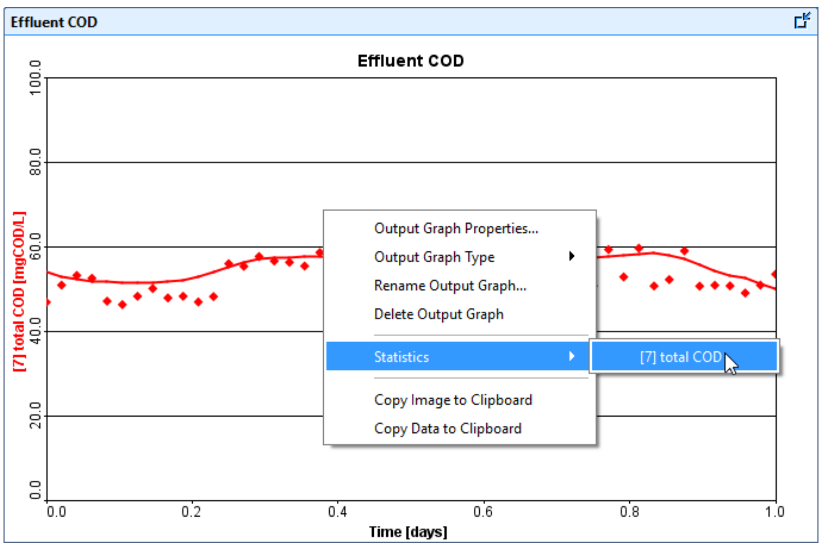


Figure 7-31 – Accessing Statistics Menu

This will bring up a menu of the available parameters on this X-Y graph as shown above. Select the desired variable to access the Statistics dialog for that particular variable.

From the Statistics dialog, you can create a table of goodness-of-fit statistics, a plot of simulated vs. measured data points, and/or a histogram of standardized residuals. Each graph will be created in a new window on the current tab, and can be auto arranged, resized and/or dragged to another tab as needed.

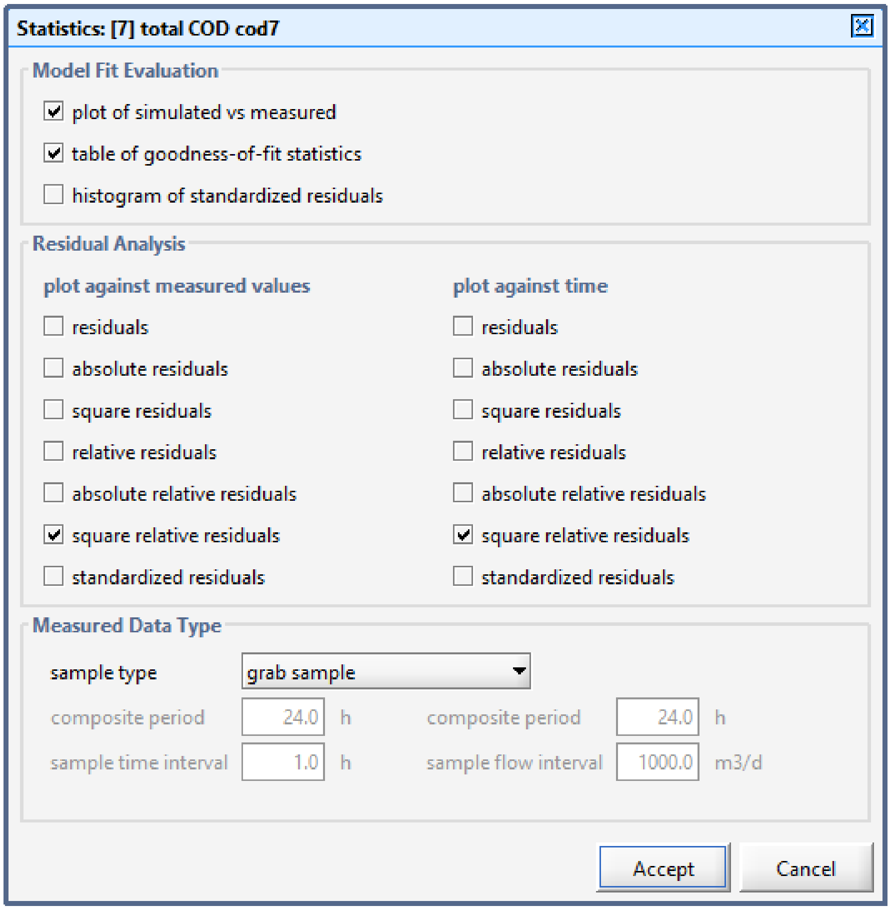


Figure 7-32 – Statistics Dialog

In addition, you can create graphs of the various types of residual analyses by selecting the desired graph from each of the two columns (plot against measured values, plot against time) in the ***"Residual Analysis"*** section of the dialog.

You can specify the nature of the measured samples being used in the statistical analyses (e.g. residuals) by selecting the appropriate sample type from the drop-down box in the ***"Measured Data Type"*** section.

Once plots/tables have been created, you can access this dialog again by right-clicking on any of the produced graphs, and selecting the **Statistics** menu option.

Full details on the calculation of all statistical analyses can be found in the ***Technical Reference*** manual.

Exporting Data from Statistical Analyses

The entire graph can be exported to an Excel spreadsheet by clicking on the "**Export Data to an Excel File** menu item found in the **Export** button drop down list on the **Outputs Toolbar.** This will open a file browser where you can choose the name and location of the file.

Additionally, the entire graph can be exported to a Word document by clicking on the "**Export Tab to Word"** menu item found in the **Export** button drop down list on the **Outputs Toolbar.** This will open a file browser where you can choose the name and location of the file.

Alternatively, you can copy the data (text only) to the system clipboard by clicking the **"Copy Data to Clipboard"** menu item found in the **Export** button drop down list on the **Outputs Toolbar.** The data can then be pasted into a report or spreadsheet.

CHAPTER 8

Building and Running a Model

DYNAMIC MODELLING & SIMULATION

Dynamic modelling in wastewater treatment starts with a formal description of the relationships between variables in unit processes; that is, development of models. This task of synthesis has been one of the primary bottlenecks in the application of models to wastewater treatment. Over the years, an extensive effort has gone into formalizing models of different unit processes in wastewater treatment. These models have matured to the point where they are valuable tools for the practitioner.

Models

Models are computer representations of real systems. The real systems of interest here are the unit processes in a wastewater treatment plant, such as the aeration basin or the sedimentation tank in an activated sludge treatment system. Models in GPS-X can be classified as either mechanistic or empirical. The difference between empirical and mechanistic models is more a continuum than a rigid classification as mechanistic models often have some empirical component.

Mechanistic models (**Figure 8-1**) are based on first principles, that is, laws of physics, chemistry, and biology. Actual data are used to modify parameters in these models. For example, the law of conservation of mass is the essential building block on which dynamic equations are based in GPS-X. Data collected in the real system are used to adjust physical dimensions, rate constants and other parameters in the equations. This is a "bottom-up" approach to modelling in which the model is provided a firm foundation by natural laws. These laws help us to understand more thoroughly the behavior observed in the real system and the model. Many of the models in GPS-X are mechanistic.

Empirical models rely solely on data. In this approach, the model structure is determined by selecting from a set of general mathematical expressions such as those shown in the center of **Figure 8-2**. There is no guarantee that the selected model will be ‘correct’. Essentially, the data "speak for themselves" and ultimately dictate the form of equations - selected among a set of candidate equations on the basis of goodness-of-fit - to be used in the model.

This "top-down" approach to modelling is weaker than the mechanistic approach as the model structure is simpler and the meaning less clear. A different set of data can often result in a different model structure and we are not always certain how to interpret the meaning of parameters in the model. Nevertheless, empirical models are useful as long as their use is limited to the data ranges over which they were fitted, and in many cases, are the only choice when we have limited knowledge about the real system being modeled. For example, empirical models often give good predictions and can ultimately lead to the discovery of more general mechanistic expressions. GPS-X includes several kinds of empirical models. In these models, actual data determine which model to select and the values of parameters in the selected model.

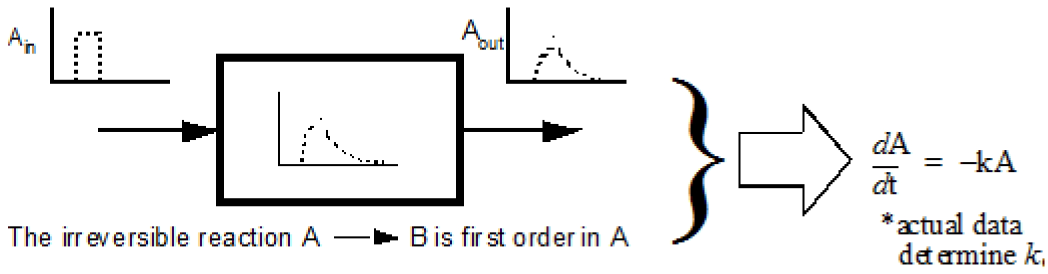


Figure 8-1 – Mechanistic Models are Based on First Principles

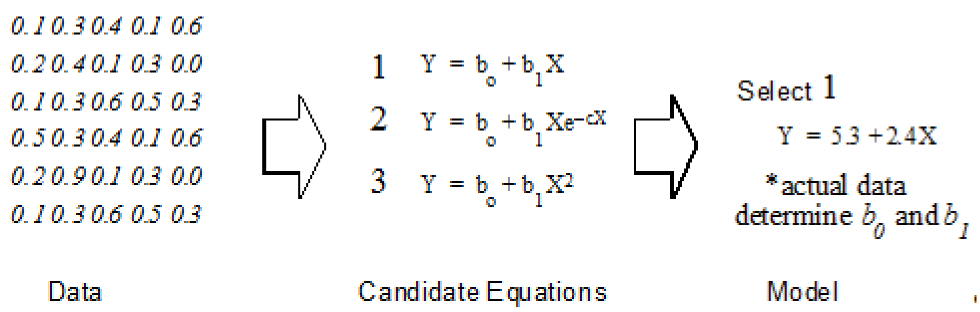


Figure 8-2 – Empirical Models Fit the Data to a Certain Model Form

State and Composite Variables

The system of differential equations, which GPS-X automatically generates in the model building process, constitutes a precise, mathematical representation that describes the relationship between model variables. Some of these variables, referred to as **State Variables**, are important because they define the **"state"** of the system[8]. In the absence of any external excitation, the state variables determine how the system behaviour evolves over time. Secondary or **composite** variables are calculated from state variables and other constants so they always depend on how the state variables change. The difference between these two variable types is demonstrated in **Figure 8-3**.

[8] Here, "System" refers to the model, that is, the system of equations.

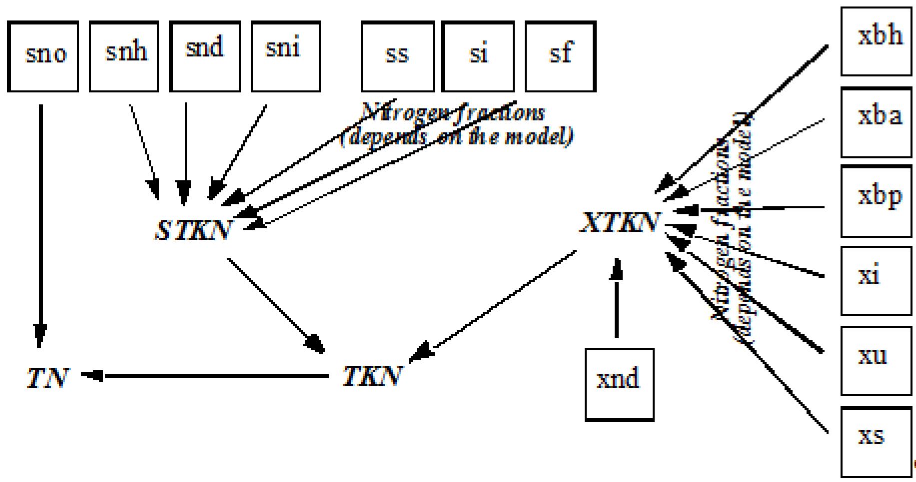


Figure 8-3 – Nitrogen State (Boxed Variables) and Composite (Bolded Text) Variables

The essential behaviour of a model is closely linked to the response of the model's state variables. This is a key to conducting effective sensitivity analyses. You can examine the behaviour of composite variables, and this is convenient as composites are often the kind of variable typically measured in a plant, for example, mixed-liquor suspended solids is a variable composed of several particulate state variables. As shown in **Figure 8-3**, Total Kjeldahl Nitrogen (TKN) is a composite variable calculated as the sum of soluble and particulate nitrogen state variables plus the nitrogen fractions of some organic state variables. State variables are the basis on which models are organized in the GPS-X libraries.

State Variables and GPS-X Model Libraries

Models are approximations of a real system. State variables are the essence of a model, but it was not described how these are selected. Given assumptions about the proper state variables, another problem is identifying the reactions that these variables undergo. This includes specific expressions for the reaction rate (kinetics) of each variable and the mass ratios (stoichiometry) between variables.

In many cases there are obvious choices for state variables, such as substrate and biomass concentrations in the activated sludge process, but in general this requires care in selection. Simplifying assumptions must be made and these are reflected in the specific choice of state variables. Modelers always try to find a balance between model complexity and prediction capability as shown in **Figure 8-4**. Increasing the complexity of a model by adding new reaction terms, etc., may increase its ability to make more accurate predictions but it carries extra overhead including more difficulty in comprehending the model structure, additional data requirements and reduced calculation speed. Removing or simplifying reaction terms minimizes these problems but may result in less accurate predictions. **No single model can explain all real system behavior under all circumstances so it is important to have flexibility in selecting the model to be used for a particular purpose**.

GPS-X gives you the necessary flexibility in model selection by providing a number of libraries containing many state-of-the-art models. The model libraries available with GPS-X are shown in **Table 8-1.**

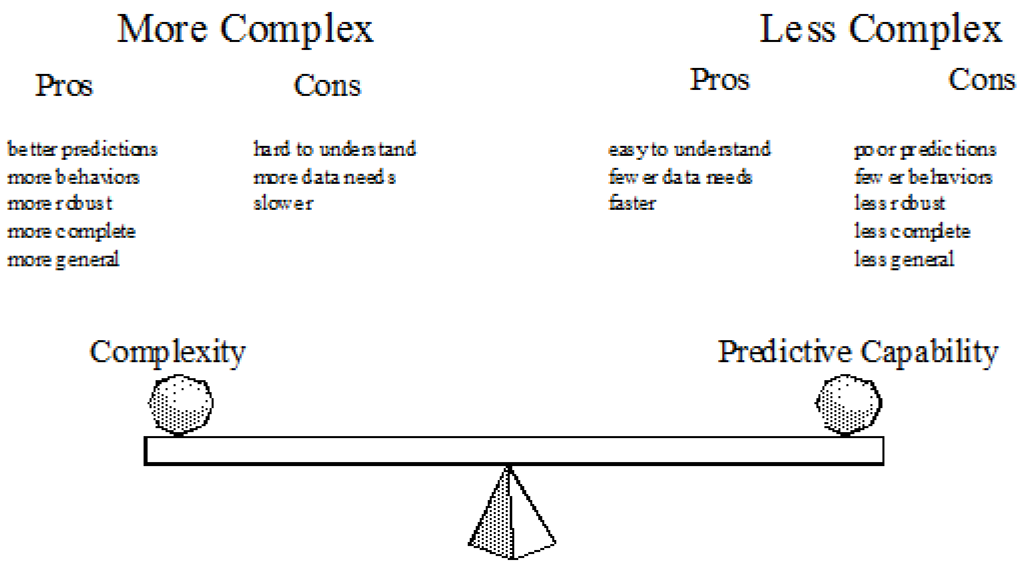


Figure 8-4 – Balancing Multiple Objects is Important in Developing a Good Model

Table 8-1 - Model Libraries Available in GPS-X

Library Name	Number of State Variables
Comprehensive (Mantis2)	52
Selenium and Sulphur (Mantis2S)	72
Greenhouse Gas/Carbon Footprint (Mantis3)	56
Process Water (Procwater)	68
Petrochemical (Mantisiw)	52
Carbon – Nitrogen (CN)	16
CN Industrial Pollutant (CNIP)	46
Carbon – Nitrogen – Phosphorus (CNP)	27
CNP Industrial Pollutant (CNPIP)	57

You have a choice with regard to the basic state variables you can model. You can get more information on each of these libraries in the ***Technical Reference***.

A Common Basis for Models

Now that you are familiar with the general nature of unit process models available in GPS-X, you may question how the GPS-X system manages the integration of different kinds of models. In GPS-X, state variables are the elements common to every model in a library. As these are the essential variables, they provide a convenient basis for combining different unit process models. All models in a library include expressions for each of the state variables. The values for these variables at entrance and exit points are particularly important because these are the points at which flows to and from other processes originate or terminate. To build a multi-unit-process model, it is necessary to track these values between unit processes.

The key task of tracking state variables between unit processes is handled transparently by GPS-X. GPS-X manages this in a rigorous way. You can display the values of these and other composite variables at every point in your plant layout, but you can ignore the procedures for mapping the state variables among the models. This is another example of how GPS-X insulates you from unnecessary detail, increasing the power of the models you are able to build and allowing you to concentrate on understanding your model and your system rather than code implementation.

Material Balance Expression

One of the key tools of practitioners in engineering is the material balance. Construction of dynamic models is a straightforward application of this fundamental principle. When GPS-X builds a model of a layout, it does so by first establishing the material balance for each state variable in the unit process. As discussed above, this involves tracking the state variables in each unit process in the layout. This is the essential task of the GPS-X translator, which converts the graphical layout into code as described in the ***Overview of The Model Building Process*** section and shown in **Figure 8-10**.

The **material balance** expression for a constant volume can be stated as:

> *The net rate of change in a component within a control volume (an object in GPS-X) is equal to the rate at which the component enters the volume, minus the rate at which it leaves the volume plus or minus the rate at which the component is generated or used within the volume.*

This is shown graphically in **Figure 8-5**. For each object in GPS-X a material balance dictates the changes which occur in the key components, that is, the state variables.[9] All other dynamic characteristics of the system depend on how these state variables change. There may be several hundred equations constructed by GPS-X if the layout is large. Both steady state and dynamic (time varying) situations can be analyzed using the same fundamental relation and the equations produced by GPS-X.

[9] The black box object (and some models in the toolbox object) in GPS-X does not use the material balance to establish input-output relationships; however, it is defined to properly map model state variables from input to output points.

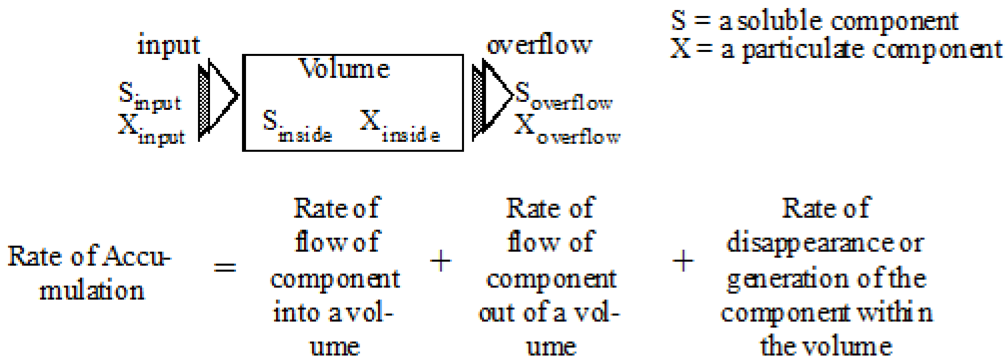


Figure 8-5 – Material Balance Expression for Two Components (State Variables) and X

Steady-State and Dynamic Simulation

From the material presented in the previous sections, you should now have a general understanding of the nature of the models in GPS-X and the way they are constructed. In the last section, it was stated that the equations can be used for analysis of both steady-state and dynamic or time-varying situations. The important differences between these types of analyses are examined this section.

Steady-State Analysis

Steady-state analysis assumes that none of the state variables change over time. To get a qualitative feel for the meaning of steady-state, assume a situation where a plant has been operating under the same external excitations for an extended period of time. The plant has been receiving the same influent, with the same composition and has been operated in the same mode for several months. The situation is shown in **Figure 8-6**. In this scenario, the influent flow and operational mode are potential sources of external excitations, but assume for the moment that they ***do not change over time***. As none of these changes, after some time, the plant settles into a steady-state. Because no external excitation is disturbing the plant, variables measured in the plant, such as DO and MLSS, do not change during the period, that is, the variables are not dependent on time. Whenever you interpret the results of a steady-state analysis, remember that the results are valid only for a plant operating at steady-state as described here.

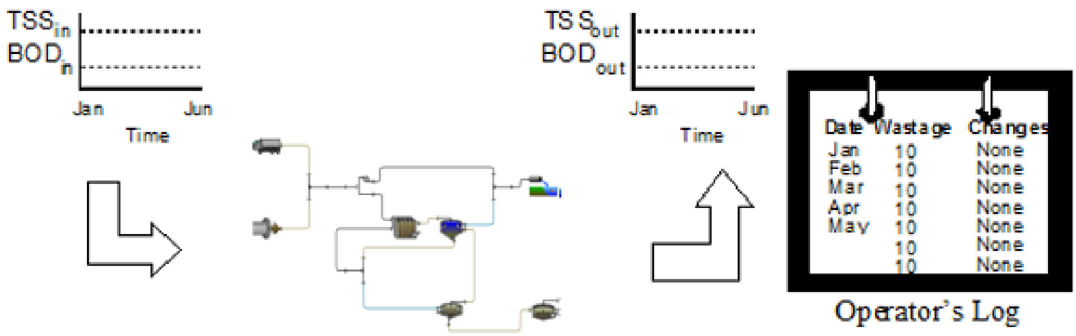


Figure 8-6 – A Treatment Plant Operating at Steady-State

Finding state variable values at steady-state involves an iterative procedure for solving a system of simultaneous equations. Normally, this procedure converges (finds the solutions) but sometimes it does not. In the latter case, the state variable values contain a (possibly high) degree of error. Ensuring that these errors are kept to a minimum is an important consideration because, if a solution cannot be found there may be a problem with the structure of the model or the parameter values being used. Whenever you conduct a steady-state analysis, make sure that the solution procedure converges to a reasonable state. The steady-state solver included with GPS-X is a robust procedure that was specifically designed for the types of models used in wastewater treatment facilities. See the ***Steady State Simulation*** section in this chapter for more information.

Conducting a steady-state analysis of a model is a convenient way to get a feeling for model variable relationships. Values calculated by the steady-state solution procedure depend on the values of all model inputs and parameters. For example, the steady-state value for effluent ammonia depends on the value for the wastage rate because this affects the growth of ammonia-removing bacteria. One quick way to see the relationship between wastage rate and effluent ammonia concentration is to conduct a steady-state sensitivity analysis. To do this, simply change the value of the wastage rate, run the GPS-X steady-state solver and observe the calculated value for effluent ammonia. Do this for several different wastage rates to see the effect over a range of wastage rates. This can be done manually or automatically if you have purchased the **Analyzer** option in GPS-X.

Dynamic Analysis

The assumption of steady-state is convenient, but most plants do not operate in steady-state. For most realistic situations, it is important to conduct dynamic sensitivity analyses to observe the time-dependence of variables. This situation is shown in **Figure 8-7**. In contrast to **Figure 8-6,** this plant is in a continuous state of change. The influent composition varies and operational changes, process upsets and equipment failures disturb the steady operation of the plant.

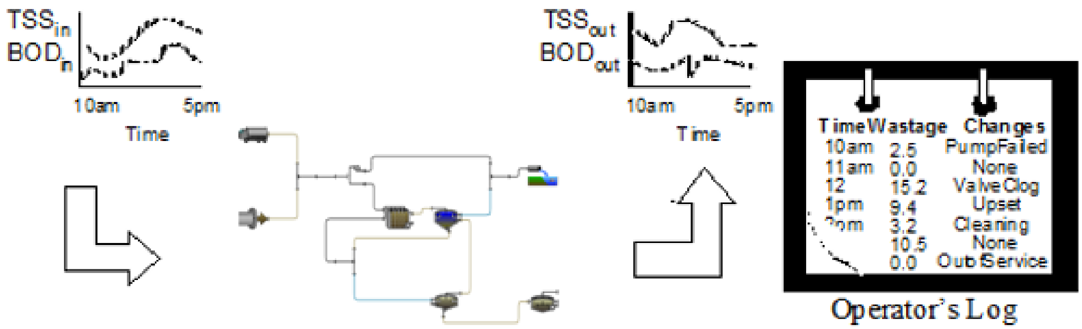


Figure 8-7 – A Treatment Plant Operating under Dynamic Conditions

Dynamic analysis gives a more complete picture and can serve to validate or reinforce the results of steady-state analysis. Most plants do not have a constant, unchanging influent, and are subject to frequent disturbances (excitations) that result in wide fluctuation in the values of measured variables. For wastewater treatment facilities, dynamic operation is the norm and steady-state operation the exception. This does not mean that results from steady-state analysis are invalid because often a dynamic analysis shows that the model variable values lie within a range, which includes the steady-state value; however, it is always a good idea to supplement a steady-state analysis with a dynamic one in cases where you suspect that the plant is significantly influenced by dynamic disturbances.

Integration Error and Instability

The benefits of dynamic simulation are significant; however, the solution of the dynamic equations is more complex and, if done correctly, can introduce errors in the results. The two major considerations in the solution of dynamic model equations are numerical instability, and the error between calculated and actual values (referred to as convergence). GPS-X is designed to minimize these types of problems. For the great majority of simulations you perform, the default conditions imposed by GPS-X will ensure adequate accuracy of the solution to the model equations. GPS-X includes robust procedures for this purpose and makes it easy for you to apply them in conducting simulations.

Dynamic analysis requires numerical integration of the model equations, that is, the *estimation* of derivatives by finite difference approximations for each of the model's state variables. When using these procedures, there is always a small error, referred to as the *integration error*, between the calculated value and the real value, which would be obtained from an exact solution of the equations. You can control this error by setting integration method parameters or by selecting a different integration method. There are several methods available in GPS-X and you can easily change from one method to another. In most cases, it is possible to keep this error below a reasonable, arbitrarily selected limit.

The second consideration that sometimes arises in numerical simulation is instability. Under some circumstances it is possible for integration procedures to become unstable. The effects of this may be minor, such as small, rapid changes in the value of a variable, or they can be dramatic, resulting in the invalidation of a simulation run. These problems typically occur when using one of the less robust integration routines, especially if the integration step size is made too large. This may be done for convenience or to save time as the large step size requires fewer calculations and therefore, results in faster execution times.

A simple way to determine if errors or instability are a problem is to run the same simulation with more than one of the available integration methods. If you find significant differences in the results of these runs, then one of the methods is producing incorrect results. Try making adjustments to the integration parameters and re-run the simulations until you get agreement between the results calculated by different integration procedures. Consult the ***GPS-X Technical Reference*** for more information on this topic.

Initial Conditions

Finding solutions to the dynamic equations, which comprise the models you prepare with GPS-X, is referred to as an **initial value** problem. In general, the behaviour of a dynamic model depends on:

1. The model equations,
2. The values of parameters and inputs to the model equations,
3. Inputs to the model (forcing functions), and
4. The initial values of the model state variables.

Without assigning initial values to the variables, it is not possible to find a unique solution to the set of differential equations that comprise the model. In previous material we have described how GPS-X constructs the model equations and the ways for you to enter model parameter values. The remaining task is to select the initial conditions. As the behavior you observe is dependent on these values, it is useful to take time to consider how initial values can be determined and how the results you obtain should be interpreted.

In all of the simulations you perform with GPS-X, it will be necessary to specify the initial conditions. There are two ways to do this:

1. Enter the values directly, and
2. Calculate the steady-state values at time=0 and use these as the initial values.

The procedures for setting the values directly have already been covered in the ***Process Data Menu*** sections. As initial values are a kind of model attribute, their values can be set from the object process data menu. The hierarchical menu obtained by selecting this item is shown in **Figure 8-8**.

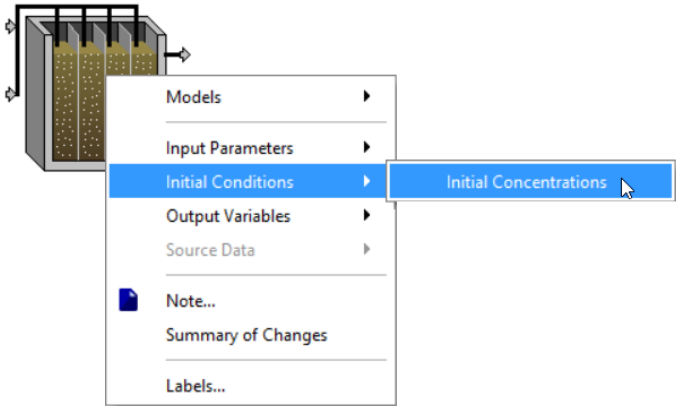


Figure 8-8 – Initial Conditions Menu Item

Remember that the initial values determine the unique solution obtained for the set of defining (differential) equations. If these values are set directly, then you must take care in making a proper interpretation of the meaning of the simulation results obtained. A safe way to set the initial values, one with a simple interpretation, is to use the steady-state solver to calculate the steady-state values and then use these as the initial values.

As shown in **Figure 8-9** there are actually two cases to consider. In case 1, shown on the left-hand side of **Figure 8-9**, the process has been at a steady-state prior to the start of the simulation (t=0). After starting, the changes made to model inputs or parameters cause some dynamic behavior to occur. This is the case mentioned above, where the steady-state solver has been used first to calculate the steady-state. After doing so, the state variables are given these steady-state values at the starting time (t=0). The state variables at any time after the start are influenced by changes to model inputs or parameters resulting in the dynamic behavior you observe.

In the second case shown on the right-hand side of **Figure 8-9**, the behavior of the state variables is dynamic before and after the point where time=0. Consider two additional scenarios of Case 2 that may be simulated; one in which there is *no* knowledge of the dynamics before time=0, and one in which there is knowledge of the dynamics before time=0.

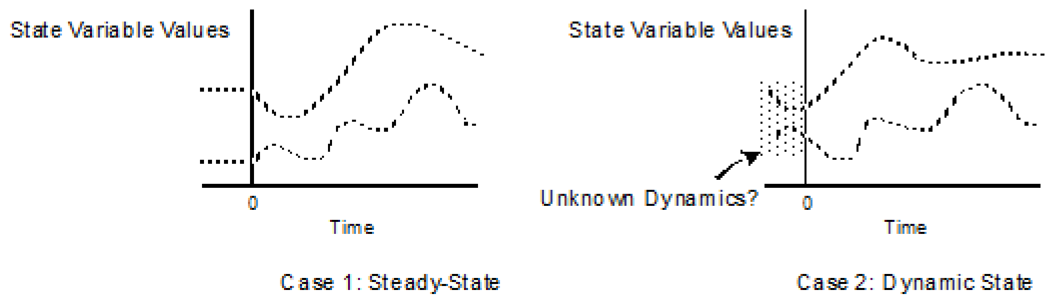


Figure 8-9 – Two Possible Simulation Cases and their Relationship to Initial Values

If there is no knowledge of the dynamic behavior before time=0 and the initial values are entered directly, perhaps from actual measurements or based on engineering judgment, then it is not always clear whether the behavior after time=0 accurately reflects the dynamics in the system being modeled. Relatively small errors in a single state variable initial condition can cause significant errors in the simulation results.

It is also possible to specify a set of initial conditions that violate model validity constraints, for example, entering a negative value for concentration or an unreasonably large or small value for a state variable. Under these circumstances the integration of model equations may still be stable with a low integration error but the results are not valid. In general, you should exercise care in the selection of initial conditions for this case.

Given good estimates of the initial conditions, the system of equations will eventually "settle in" to a repeatable pattern. This period can be estimated as it is dependent upon the time constants for the processes being modelled. In wastewater treatment processes, important time constants are the hydraulic detention times of the unit process and the sludge age or solids retention time (interpreted as the mean age of the solid matter) of a system of unit processes, for example, the activated sludge process. Generally, a value of 2 to 3 times the hydraulic or solids retention time should be used as a minimum period.

If there is knowledge of the dynamics before time=0 then it is possible to determine exactly the derivatives and, therefore, the behavior of the system. This would be the case if you restored the initial conditions from a previous run or when there is a repeating pattern of model inputs or parameter changes, such as a repeating diurnal variation in influent flow rate and composition. For the latter case, you may at first only have estimates of the initial conditions but you can determine the exact initial conditions by re-running the simulation with the same inputs until you get identical results for consecutive runs. GPS-X includes this capability and will perform the necessary *re-runs* automatically.

You have many options in setting-up and running simulations. In some cases, it is not necessary to run dynamic simulations because you can get the information needed by completing a steady-state analysis. In other cases, extensive dynamic analyses are necessary. For example, a simple treatment plant design might require only steady-state values, whereas an on-line application may necessitate closer attention to process dynamics. Take some time to think about the results obtained to ensure that the simulation results are understood correctly. GPS-X helps you to get the simulations right - interpreting correctly is up to you.

OVERVIEW OF THE MODEL BUILDING PROCESS

The *build* procedure is shown in **Figure 8-10**. In GPS-X descriptive models are supplied in libraries. When you place objects on the drawing board and specify their **Model** types, GPS-X knows which models to retrieve from the library. When you specify the connections between objects, GPS-X then knows how to couple these models to generate a dynamic model of the entire layout. As default data are defined for all critical model parameters, it is possible to translate the layout into a high-level computer language and generate an executable program.

In GPS-X, the model build process is transparent; once you have prepared the layout on the drawing board, creating an executable model is done when switching over to **Simulation Mode**. GPS-X achieves this using a specialized translator, which converts the graphic images you place on the drawing board into dynamic model equations. Once the executable code is generated, you can conduct simulations of the model.

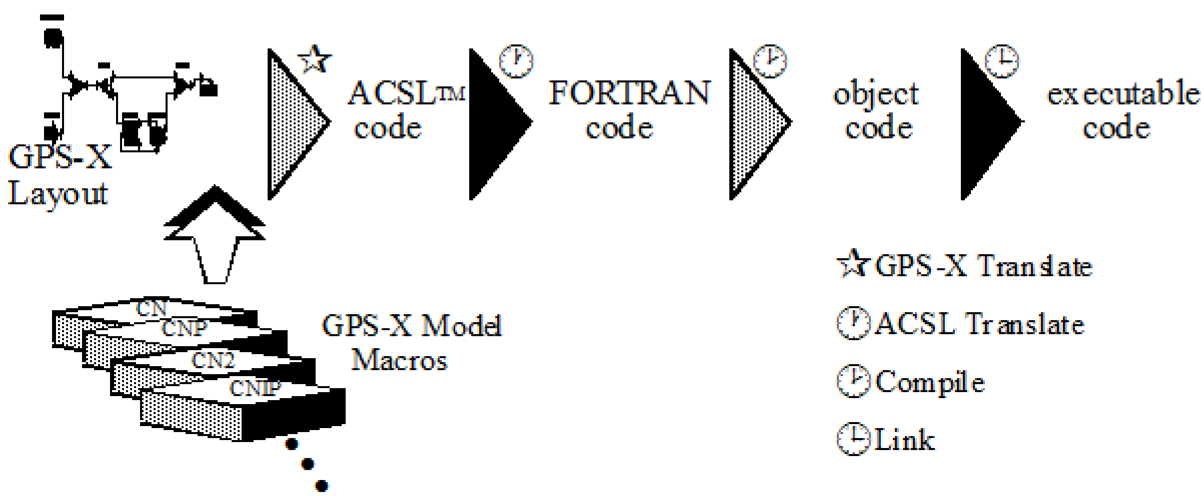


Figure 8-10 – The Model "Build" Process in GPS-X

As shown in **Figure 8-10**, GPS-X simplifies the construction of models by automating many tasks with special routines. The first step involves the GPS-X translator, which is a special pre-processor program that converts the layout you prepare to a high-level description of a dynamic model. The GPS-X translator writes the material balance equations for each unit process based on the models and connectivity you specified. Many levels of detail, which concern the mechanics of constructing the code rather than process understanding, are hidden through the use of an extensive macro language. This allows you to concentrate your efforts on understanding the model itself rather than the modelling and simulation tools.

The remaining steps in the build process do not require user interaction. The GPS-X translator writes a high-level description of the model in the Advanced Continuous Simulation Language (ACSL™). ACSL is a third-party program that is used as the simulator module in GPS-X. ACSL handles the details of numerical simulation, including the ordering of equations, integration and many other details. In the next step, ACSL code is converted directly to a FORTRAN program, then to object code and finally to an executable module. The entire build process can take less than a minute to complete, even for moderate-sized models. When you run a model in GPS-X, you are running the executable module shown in **Figure 8-10**.

BUILDING A MODEL

When switching from Modelling Mode to Simulation Mode (see **Modelling/Simulation Mode** section in CHAPTER 1), if necessary, the numerical model will be built.

If the model for this layout has previously been built, but the layout has been modified, you will be prompted to rebuild the layout or simulate with the older version of the model (in most cases, you will want to rebuild the model).

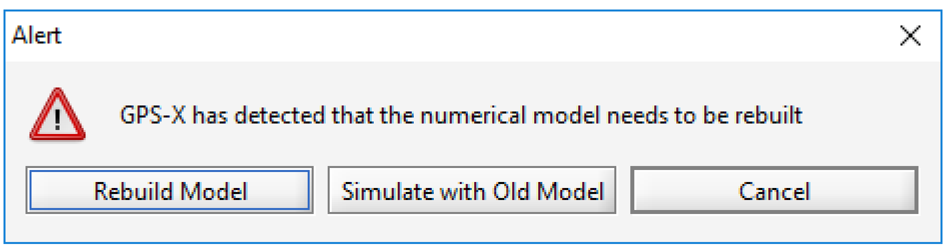


Figure 8-11 – Rebuild Model Option

When the model is being built, you will see the "**Building Model**" window, as shown in **Figure 8-12**. The status of the building process will be updated as it moves through the different steps and any errors that occur will be displayed in this window.

Building the numerical model may take from 5 seconds up to a couple of minutes, depending on the speed of your computer and the complexity of your model. When the model building is complete, the "Building Model" window will automatically close, and the model will be ready for simulation.

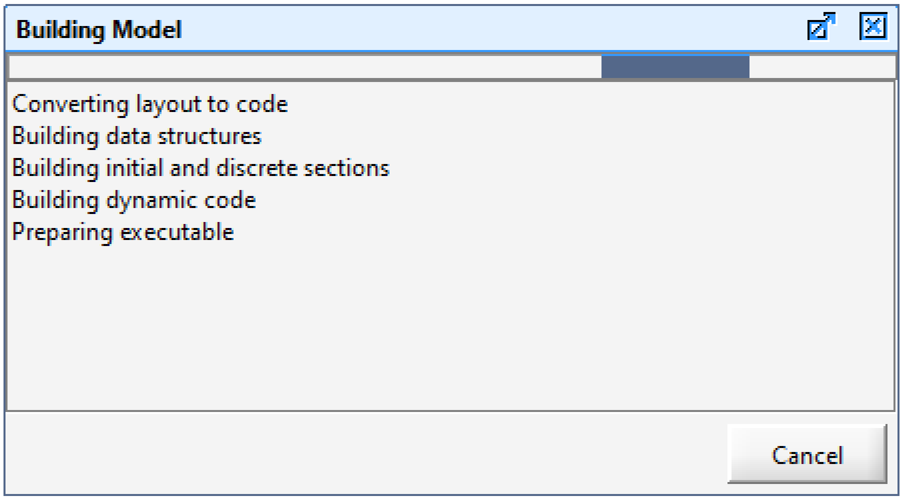


Figure 8-12 – Building Model Dialog

BUILDING OPTIONS

NOTE: This section is for advanced users. Unless you are interested in the details of the model building process, you can skip forward to the section on **Starting Simulations**.

The build process, while fully automated, can also be carried out manually from the ***Tools > Build*** menu while in **Modelling Mode** for those situations where you may want to test whether a model will build without an error but you are not ready to move on to running simulations yet.

The process of building involves two major steps:

1. Translating the drawing board layout to ASCL source code.

2. Compiling and linking the source code to produce a FORTRAN executable program.

All messages related to the build process will be displayed in the **Building Model** window (**Figure 8-12**). This includes the GPS-X and ACSL translator messages, compiler messages and linker messages.

If the build process is successful the window is automatically closed. Otherwise, it will remain so that you can view what the problem is. Some of these errors can be solved by adjusting the Translation Options.

Translation Options

If you wish to control how the translation step is performed, you can specify building details in the **Build** tab of the ***View > Preference*** menu, as shown below.

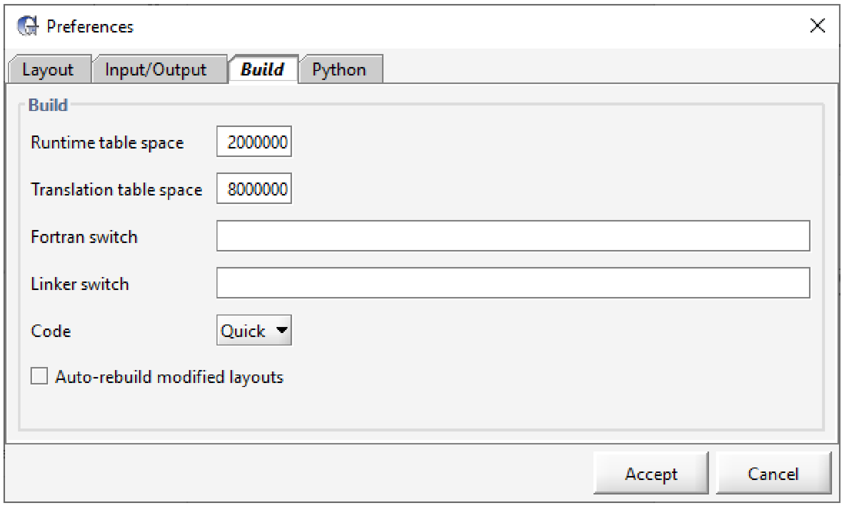


Figure 8-13 – Preferences Dialog (Build Tab)

The **Code** item has several options for the code generation step. The **Quick** option creates code optimized for speed whereas the **Big** and **Big+** options are used for large treatment process models. These options are described in greater detail below. If you make changes to the code generation options, you must subsequently select the ***Tools > Build*** menu item to prepare the executable program.

QUICK Option

Many GPS-X models contain recycle streams between processes. To find an accurate solution to the model equations in these cases, the numerical solution routines must iterate at every time step to resolve the effects of changes occurring in upstream units or downstream units, which in turn, may affect upstream units via recycle streams.

At any point receiving a recycle, the (numerically integrated) components of the recycle stream cannot be known accurately until effects are propagated through the plant to the source of the recycle. At some point, a balance is struck, but finding this balance requires an iterative solution. The **Quick** option ignores these effects and the computational overhead associated with them. Using the **Quick** option effectively reduces the execution times.

The difference in model predictions made using the **Quick** option and the exact solution are usually not significant. This is because integration time steps are much smaller than the time constants of processes in a typical wastewater treatment plant and the changes, which occur between time steps, are small. For most purposes, the **Quick** option is the best choice. The **Quick** option is assumed if you select the **Build** menu option.

BIG Option

When building a large plant model containing many unit processes (for example more than 10 settlers) it is possible to exceed the limits which the simulator module places on the number of discretely executed code sections.

When this occurs, the translation to source code will fail and an error message (‘*too many discretes, build BIG*’) will be displayed in the **Building Model** window. In most circumstances, GPS-X will auto-detect when a build fails with a “too many discretes error”, and automatically rebuilds the layout with the **BIG** option.

When the **BIG** option is used, all controllers in the layout use the same discrete sampling interval, rather than individual sampling intervals specified in each object. This **controller sampling time** is specified in the ***Layout > General Data > System > Inputs Parameters > Dynamic Solver Settings*** menu.

BIG+ Option

This option is similar to the **BIG** option. Instead of defining a single sampling interval for all controllers, scalar PID controllers (only) are disabled.

STARTING SIMULATIONS

GPS-X allows you to conduct simulations interactively rather than (or in addition to) batch model simulations. Interactive simulation is simple – you **Start** the model and then change the pre-defined controls to observe the response of model variables set up in output displays. There are some useful set-up commands, which you may want to issue before beginning and you can interrupt the simulation at any time, query the simulator for specific variable values, modify the displays or controls and then continue with the simulation. GPS-X is designed to make this interaction easy in order to get the maximum amount of information from a simulation run.

Steady State Simulation

To start a steady-state simulation:

1. Make sure you are in **Simulation Mode**
2. Select **Steady State** on the **Simulation Toolbar**.
3. Enter a value of 0.0 for the **Stop Time**.

4. Click on the **Start** button.

The simulator will then run the steady state solver and pause when it has reached convergence. The percentage convergence is shown dynamically on the progress bar on the **Simulation Toolbar**.

If you have the **Command Window** displayed, the status of the steady state calculations is displayed dynamically. This information consists of the following: the iteration number, the number of loops without improvement, the current value of the sum of derivatives and the lowest value of the sum of derivatives are shown. Messages indicating whether convergence was or was not achieved are also displayed in this window.

In some cases, the steady state solver does not converge; however, simply re-running the solver may result in convergence. This is done automatically in GPS-X if the number of retries is set to a value greater than zero (0).

To set the number of steady state solver retries:

1. Switch to **Modelling Mode**
2. Select the ***Layout > General Data > System > Input Parameters > Steady-State Solver Settings*** menu item. The steady-state solver parameters dialog is displayed as shown in **Figure 8-14**.
3. Change the value of **Number of Retries on Iteration**. A value of 5 or less is best.
4. 'Accept' to close the dialog.
5. Switch to **Simulation Mode**. You will be prompted about rebuilding – select "**Rebuild Model**". This will create a new model with 5 (or fewer if convergence is achieved) steady-state retries as default.

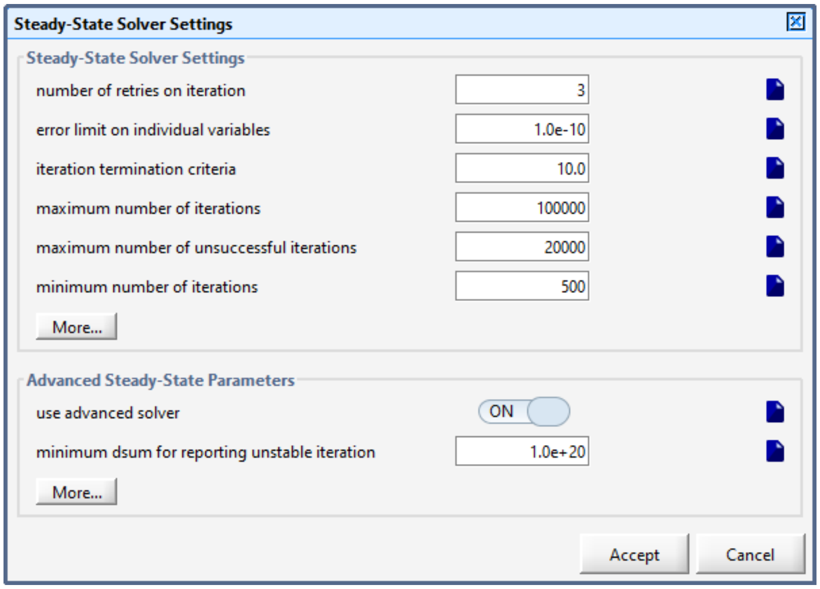


Figure 8-14 – Steady-State Solver Settings Dialog

An alternative procedure would be to set the number of retries in a scenario rather than changing the layout. This method does not require re-compilation (see ***Using Scenarios*** section of this chapter for more details).

GPS-X will re-initialize the steady-state solver and attempt to find a solution to the equations. The number of retries should be kept small as it is unlikely that the system will converge if the procedure has failed after 5 retries. Under these circumstances, it still may be possible to achieve convergence by modifying other solver parameters shown in **Figure 8-14**. As a last resort, it may be necessary to modify either the model structure or model parameters to achieve convergence.

Refer to the ***Technical Reference*** for more information on the steady-state solver and strategies for modifying steady-state solver parameters.

If the stop time is set to 0.0, the steady-state solver stops after calculating the steady-state values. If the stop time is set to a value greater than zero (0), the simulator sets the initial conditions equal to the steady-state values and starts a dynamic simulation.

Dynamic Simulation

When performing a dynamic simulation, the simulator will always stop when it gets to the indicated **Stop Time** value; therefore, before starting a simulation, make sure the current value of **Stop Time** is greater than zero (0).

To start a dynamic simulation:

1. Make sure you are in **Simulation Mode**
2. Select **Steady State** on the **Simulation Toolbar** (optional).
3. Enter a value greater than zero for the Stop Time.

4. Click on the **Start** button.

As discussed in the ***Initial Conditions*** section in CHAPTER 4, the initial conditions can be set either by entering the values directly or by calculating the steady-state values and using these as the initial conditions.

When steady-state is selected, the steady-state solver starts and the state variables, values calculated by the solver, are used as the initial conditions.

If steady-state is not selected, the simulator takes the pre-set initial conditions[10] and immediately begins a dynamic simulation run.

In the former case, there will be a delay before the results begin appearing on output displays as the steady-state solver runs.

The simulator will continue until it reaches the stopping time.

In the ***Initial Conditions*** section, problems associated with using estimates for the initial conditions were addressed. The best situation is one in which a repeating pattern of model inputs can be assumed. In these cases, it may be necessary to run the same simulation several times before the system reaches a dynamic, or periodic, steady-state. These additional simulations are performed automatically in GPS-X if the number of *re-runs* is set to a value greater than zero (0).

To set the number of dynamic simulation re-runs:

1. Switch to **Modelling Mode**
2. Select the ***Layout*** > ***General Data*** > ***System*** > ***Input Parameters*** > ***Simulation Run Settings*** menu item. The dialog window is displayed as shown in **Figure 8-15**.
3. Change the value of **Number of Reruns**.
4. ‘Accept’ to close the dialog
5. Switch to **Simulation Mode**. You will be prompted about rebuilding.

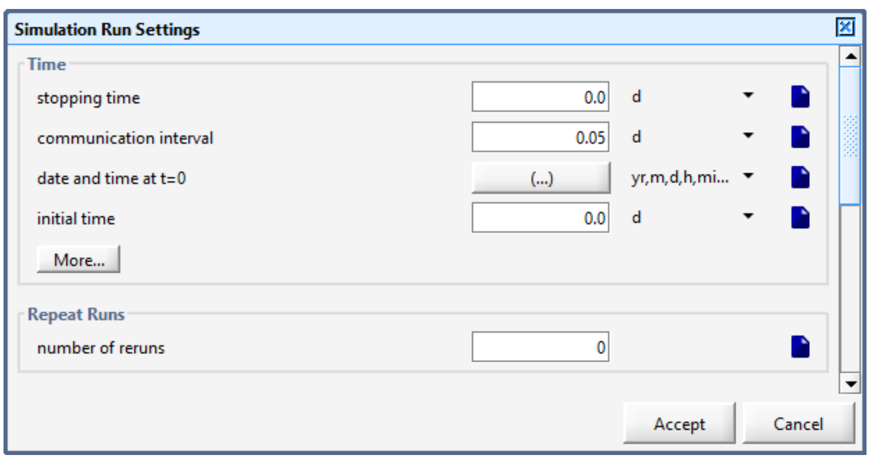


Figure 8-15 – Simulation Run Settings Dialog (Repeat Runs)

In most cases, Steps 2-6 are made in a scenario rather than the layout (see ***Using Scenarios*** section of this chapter). The number of re-runs you select will depend on estimates of the time needed for the system to come to a periodic steady-state. This

[10] Initial conditions are process object attributes and can be accessed from the object’s process data menu

estimate can be based on important time constants in your system, for example, the detention times of unit processes or solids retention times in combinations of unit processes such as the activated sludge process.

PAUSING/RESUMING THE SIMULATION

Whenever the simulator is running, you can pause the simulation by pressing the **Pause** button on the **Simulation Toolbar**.

Once the simulator paused, you can adjust the interactive input controls, issue commands, alter the integration attributes, etc. After making any changes, you can continue the simulation.

To resume a simulation when the simulator has been paused, press the **Resume** button. The simulation will continue from the point at which it was previously interrupted.

You can pause and resume as many times as you want during a simulation session.

OUTPUT VARIABLE FORM

Normally the variables of interest are displayed in graphs which are defined by the user (e.g. bar graphs, XY graphs, etc.); however, at times it is convenient to view the values of variables that were not included in these displays.

One useful feature for quickly checking the values of certain variables is the ability to browse the output variable forms of the various unit processes in your layout.

1. Make sure you are in **Simulation Mode**
2. **Right-click on the object** of interest on the drawing board. The object's process data pop-up menu will be displayed.
3. **Select the 'Output Variables' item** and browse the submenu system to find the category of interest. Select the item and the Output Variable Form will be displayed. The current value of the variables are displayed on the form.

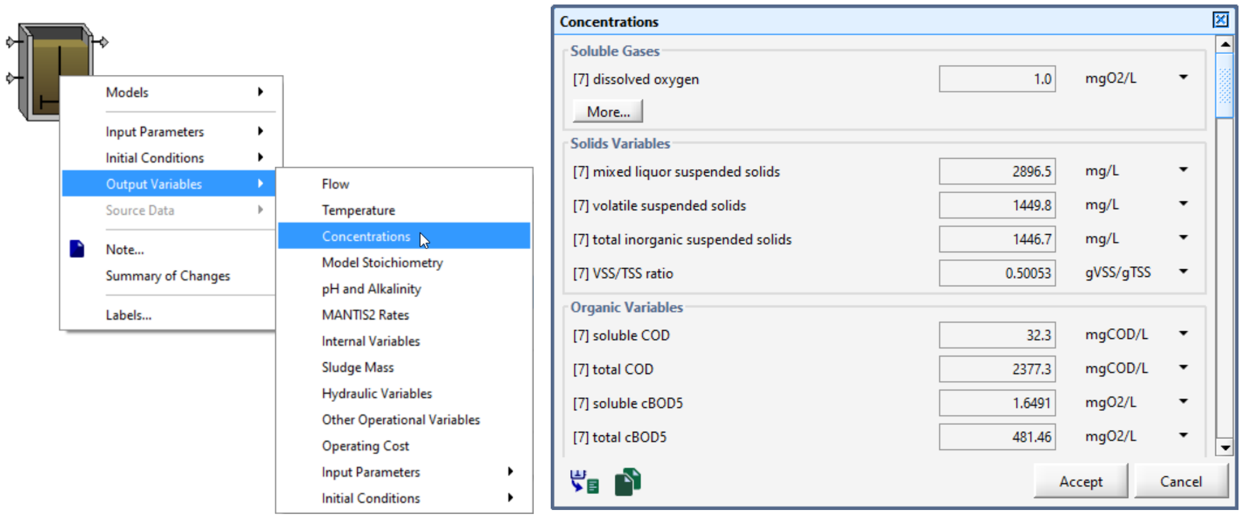
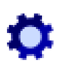


Figure 8-16 – Accessing Output Variable Forms

SIMULATION CONTROL



On the **Simulation Toolbar** there is a button to access various settings regarding the simulation and some different low-level commands that can be sent to the model.

Integration Control

This item allows you access to the window that contains relevant information for the internal numerical solver. Note that these values can only be altered in a scenario (see **Using Scenarios**) or in **Modelling Mode** by accessing the same parameters through ***Layout > General Data > System > Input Parameters > Dynamic Solver Settings***.

Properties

This item allows you access to the settings (ie. min/max/delta) for the **Stop, Communication** and **Delay** data entry fields.

Commands (Advanced Users)

When in Simulation Mode, and whenever the simulator is in the idle state, it is possible to query the simulator directly by issuing ACSL runtime commands. These commands can be used to exercise the model, that is, to run a simulation, look at the results, and change model constants.

Most of the essential commands and coordination with the simulator are handled by the GPS-X interface; however, you have the ability to issue ACSL commands directly.

A full description of the ACSL commands is given in the *ACSL Reference Manual* (contact Hydromantis for details).

NOTE: Commands issued to the simulator are executed in the order given and data values that are set remain the same until they are changed by a subsequent command or the model is unloaded. These commands can conflict with GPS-X instructions to the simulator. For example, if you set the value of a model parameter in a GPS-X process data entry form and then later use the runtime command ***"set"*** to set the value of the same variable, the latter action will override the former and the value displayed in the GPS-X data entry form will be incorrect.

Eight commonly used commands or procedures[11] are pre-defined and provided in the ***Simulation Control > Commands*** drop down menu on the **Simulation Toolbar**, as shown in **Figure 8-17**.

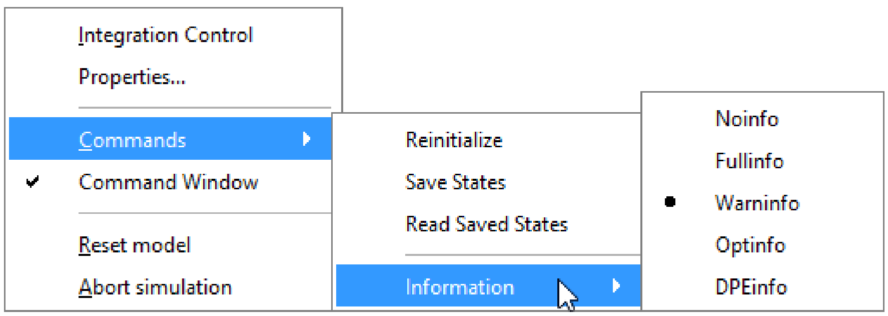


Figure 8-17 – Simulation Control (Commands Submenu)

Five of these procedures are accessed from the **Information** sub-menu. They concern information messages that are sent to the **Command Window**.

These procedures are:

1. **Noinfo** – suppresses the display of GPS-X system and simulator module messages.
2. **Fullinfo** – displays all messages.
3. **Warninfo** – suppresses the simulator module messages, but displays GPS-X system messages.
4. **Optinfo** – sets up the **Command Window** for display of optimization information in optimize mode (see CHAPTER 10).
5. **DPEinfo** – sets up the **Command Window** for display of dynamic parameter estimation information (see CHAPTER 10).

[11] Procedures are a series of ACSL commands

Information on the messages that are displayed can be found in the ***Technical Reference***.

The remaining pre-defined commands in the ***Simulation Control > Commands*** menu are provided for saving and restoring or reinitializing the model state variables.

1. **Reinitialize** procedure writes the current values of the state variables into their corresponding initial conditions. This is useful for establishing a starting point for a subsequent simulation run. This procedure stores these initial conditions in memory only. It does not overwrite the values that a user sees in the initialization forms, so if the model is unloaded and reloaded, the form values will be used as the initial conditions. The reinitialized values will be lost when the model is unloaded.

2. **Save States** procedure saves all model data, including state variable initial conditions to a data file. This file has a pre-determined name (ie. ACSLSV.gps) and is located in the same directory as the layout file, so only one set of model data can be saved at a time using this feature.

3. **Read Saved States** procedure restores the file output created using the **Save States** command. This is another useful way to save the data at a point in the simulation so that you can later start a simulation at the same point

Any messages or other output from the simulator module will be displayed in the Command Window.

Command Window

The **Command Window** is an auxiliary output area used to report details of the system and model status. These include messages that might be generated during run-time.

In addition, the **Command Window** is the primary text-based output interface to the simulator module.

You can issue commands to the simulator (as described in the section above) or any other valid ACSL command (see ***Technical Reference*** for more details), and see the responses in the **Command Window**.

To display the **Command Window**:

1. Switch to **Simulation Mode** (if not already there).

2. Click on the Simulation Control button on the **Simulation Toolbar**, and select **Command Window**. The command window will be displayed as a tab in the Output section of the main window.

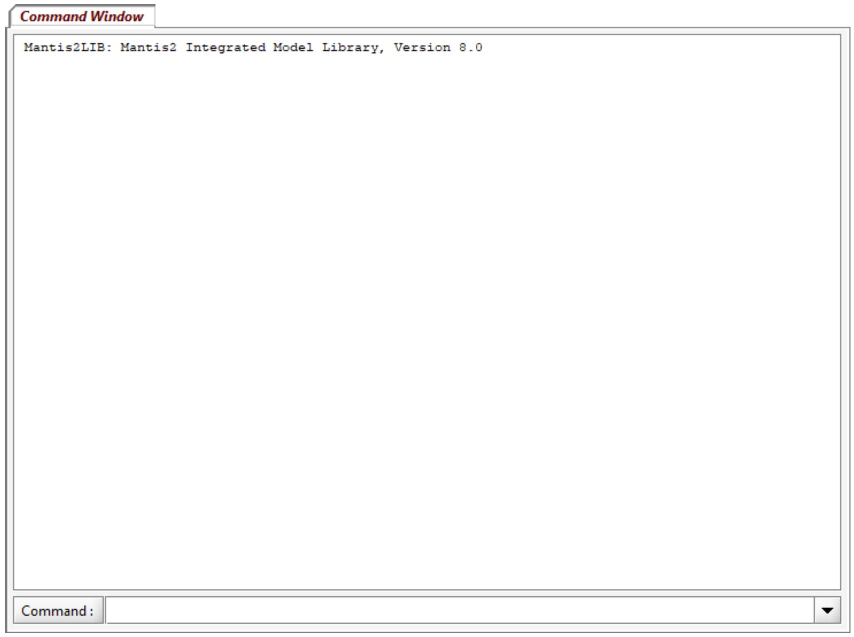


Figure 8-18 – Command Window Tab

The ACSL commands can be typed in the field at the bottom of the window (see **Figure 8-18**) and sent by either pressing 'Enter' or clicking the 'Command' button.

USING SCENARIOS

When organizing simulation runs it is useful to start with a base set of data, and then create one or more separate cases, which are modifications to the base data set. These cases are referred to as *scenarios* in GPS-X.

You can create any number of scenarios and in each scenario, you can specify the changes to the model parameter(s) which define that scenario. Those changes are saved so that they can be restored at any point in the future.

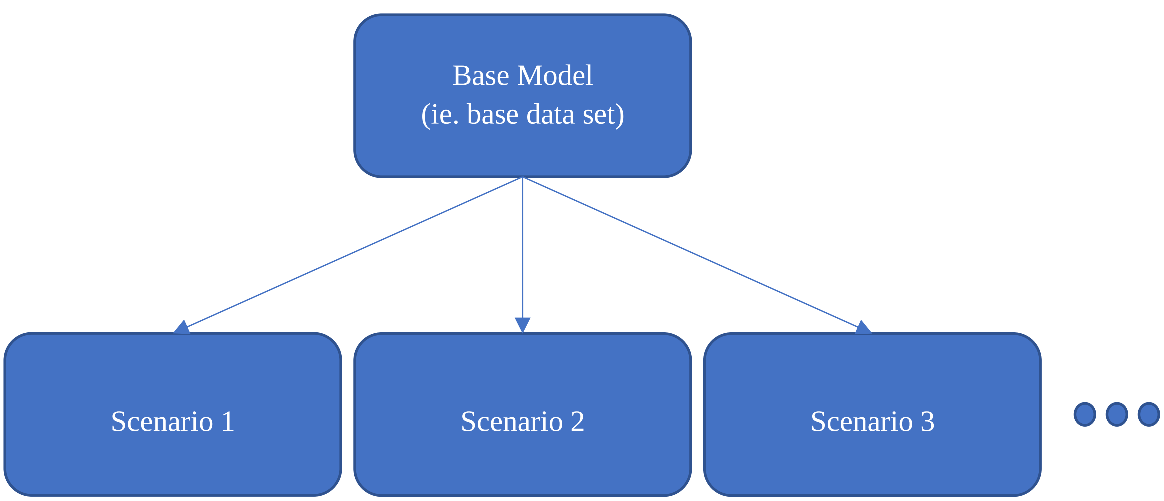


Figure 8-19 Scenarios Derived from a Base Data Set

A couple of rules that apply to scenarios:

- Scenarios can only be selected at the start of a simulation. This means that if you pause a simulation and then change the scenario, the simulation will no longer be able to be resumed. It will have to start from the beginning again.
- Changes to model parameters which would require rebuilding the model, cannot be placed in a scenario. For example, it is not possible within a scenario to change the number of reactors in a plug-flow tank, the number of layers in a sedimentation tank or delete unit process objects. You can, however, set the volume of a tank to zero to make it appear that the tank is not present.

The Scenario menu can be accessed through the Scenario item on the **Simulation Toolbar.**

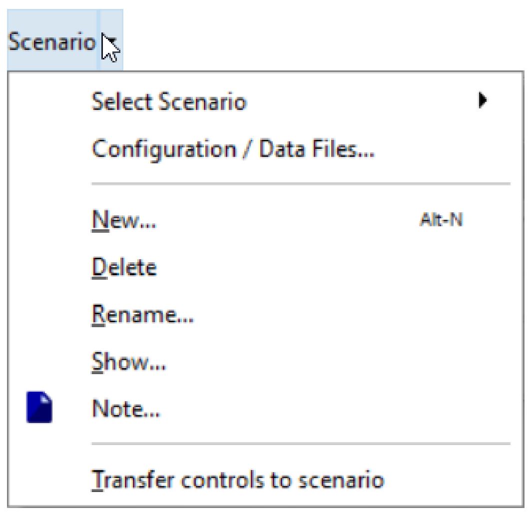


Figure 8-20 – Scenario Menu

Creating a New Scenario

To create a scenario:

1. Make sure you are in **Simulation Mode**
2. Click on the **Scenario** item on the **Simulation Toolbar.**
3. Select **New**. A dialog window will be displayed that allows you to select either the default scenario or another existing scenario to derive the new scenario from. If you select the default scenario, then you are starting from the base case. If you select another scenario, then all of the variable changes in that scenario will be copied to the new scenario as a starting point for the new scenario.

4. Type in a scenario name. Any contiguous alphanumeric name can be used.

5. Click the **Accept** button to create the scenario. This will also make it the active scenario.

Now you can go to the drawing board and access any of the object's process data menus to make the desired changes to the parameters. The title bar of the data entry form contains the message – **SIMULATION IS LOADED** – to indicate that you are in a simulation mode, and that any changes made will be added to the scenario.

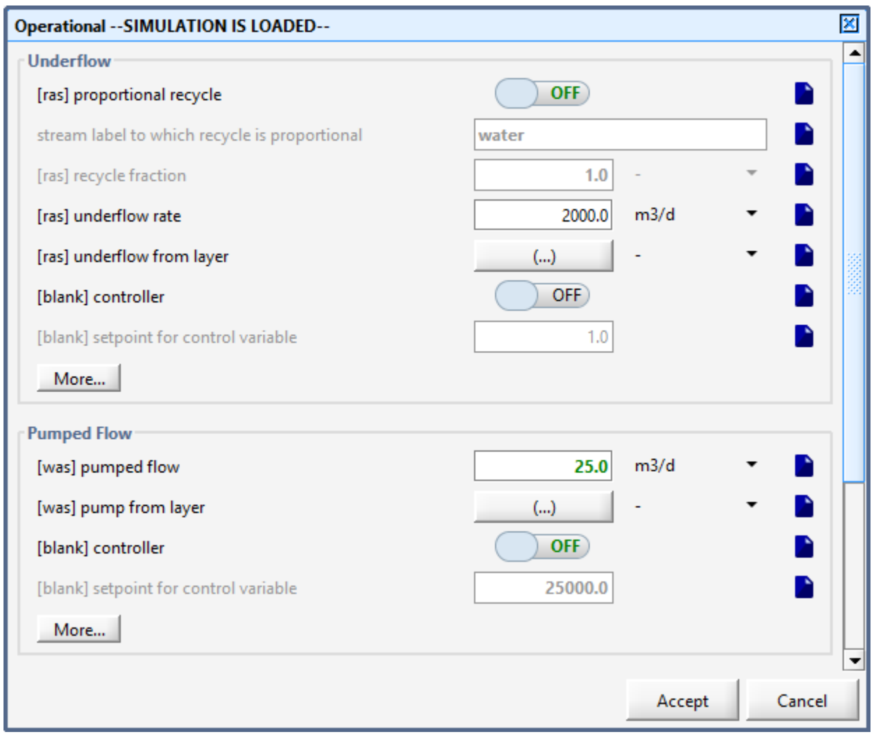


Figure 8-21 – Scenario Data Entry Form (changes shown in green)

Note that all parameter values that are changed as part of a scenario are shown in bold green text in data entry forms, for easy identification.

Selecting the Active Scenario

The currently active scenario can be selected by accessing the scenario menu on the **Simulation Toolbar** and choosing the desired scenario from the "Select Scenario" submenu.

The active scenario is displayed on the status bar below the **Simulation Toolbar.**

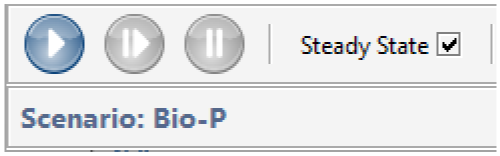


Figure 8-22 – Display of Active Scenario

Scenario Configuration

From this dialog window, you can control various aspects of the scenarios. There are four main features:

1. Add/remove/view data files. See the **Adding Input Files to a Layout** section in CHAPTER 6 for more information.

2. Comparing two or more scenarios to see the differences in tabular format.

3. Deleting scenarios.

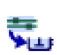
4. Reordering the list of scenarios.

Viewing Scenario Variables

On the **Simulation Toolbar**, you can use the ***Scenario* > *Show*** dialog window to view the list of variables that have been changed (from the base scenario settings) in the currently active scenario. The base scenario variable settings appear in a grayed-out box well the modified values appear in green text and can be directly manipulated from this window.

From this dialog, you can also select one or more of the variables and either:

1. **Delete** them from the scenario, or



2. **Transfer Values to layout** (ie. base data set). This means that the selected values from the scenario will be set as the base value and removed from this scenario. This requires the layout to be automatically rebuilt.

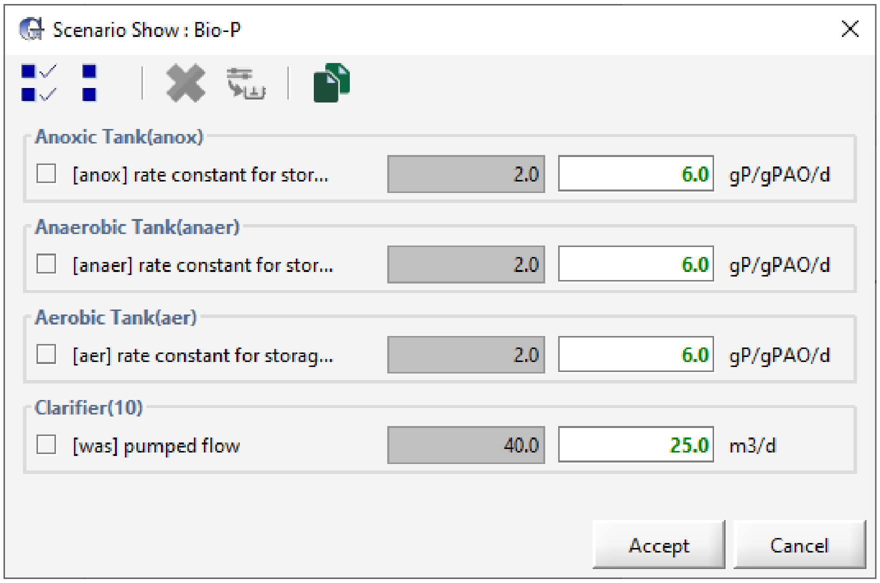


Figure 8-23 – Example of Scenario Show Dialog

AUTOMATING SIMULATIONS (AUTORUN)

The **Autorun** feature automates a number of common button click events used for running simulations. This automation is done through a text file, `<layoutname>.xec` that can be created with a text editor and saved with the other layout files. The file contains a sequence of commands which are executed in the order in which they appear in the file.

Following is a list of available commands:

Table 8-2 - .xec Commands

.xec file commands:	
log	DISPLAY LOG WINDOW
scenario <scenario name>	SET SCENARIO NAME
optimize <optimize type>!<objective function>	SET OPTIMIZE TYPE/OBJECTIVE FUNCTION Optimize type is one of: time series, probability or dpe Objective function is one of: absolute difference, relative difference, sum of squares, relative sum of squares or maximum likelihood
Analyzer <analyze type>	SET ANALYZE MODE

	Analyze type is one of: steady-state, phase dynamic or time dynamic
steady	SET STEADY STATE ON
iconize	START GPS-X IN CLOSED STATE
tstop <tstop> [d/h/m/s]	SET STOP
cint <cint> [d/h/m/s]	SET COMMUNICATION INTERVAL
delay <delay> [s]	SET COMMUNICATION DELAY
command <command>	ENTER ACSL COMMAND
start	START SIMULATION
report	GENERATE REPORT AT END OF RUN
exit	STOPS ACSL RUN, QUITS GPS-X
help	DISPLAY INFO FILE

The `.xec` file is executed by default if it exists and GPS-X is started with the `-L <layout name>` option. This feature can, however, be turned off by using the `-x 0` option when starting GPS-X.

CHAPTER 9

Analysis Tools

INTRODUCTION

The **Analyze** module is used to conduct sensitivity analyses on your layouts. This module is an optional feature. If you have not purchased this module, contact us for pricing information.

The objective of a sensitivity analysis, in the context of simulation, is to determine the sensitivity of the simulation model's output variables (*dependent* variables) to changes in its parameters (*independent* variables). This provides insight into the model's behaviour and helps identify the parameters that have the greatest impact on the model. The results of a sensitivity analysis are very useful when setting up a parameter estimation run because they allow you to determine which parameters should be adjusted by the optimizer.

You can perform sensitivity analyses using any operational, stoichiometric, kinetic, or physical model parameter as the independent variable. In addition, you can conduct both steady-state and dynamic sensitivity analyses.

The material presented in CHAPTER 8 provides a basis for some of the discussions in this chapter. In particular, the material in the **Steady-State and Dynamic Simulation** sections will be needed to understand how to set up sensitivity analyses properly.

WHAT IS STEADY-STATE ANALYSIS?

Steady-State analysis is useful when it is unnecessary to introduce the extra complexity of dynamic changes. For example, graphs such as those in **Figure 9-1**, which show the steady-state responses of model outputs to selected values of an input, might be sufficient for design purposes.

In operations, it is useful to know the ultimate or steady-state response of the plant, such as the effects of moving to a different operating point. A preliminary steady-state analysis may be sufficient to make useful predictions about the behaviour of the plant. It is important to keep in mind the proper interpretation of a steady-state analysis as described in the **Steady-State Analysis** section of CHAPTER 8. You can always enhance the analysis later by including a study of model dynamics.

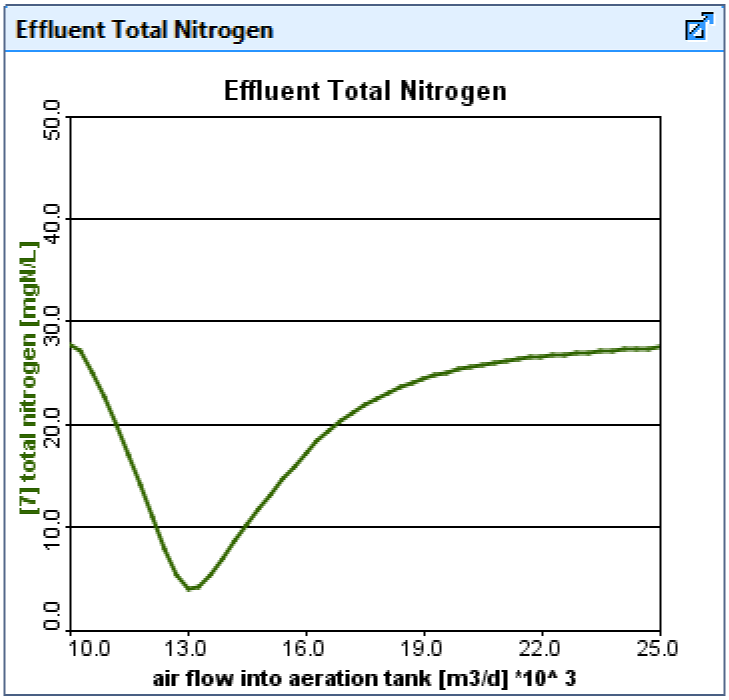


Figure 9-1 – Example of Steady-State Sensitivity Analysis Graph

WHAT IS DYNAMIC ANALYSIS?

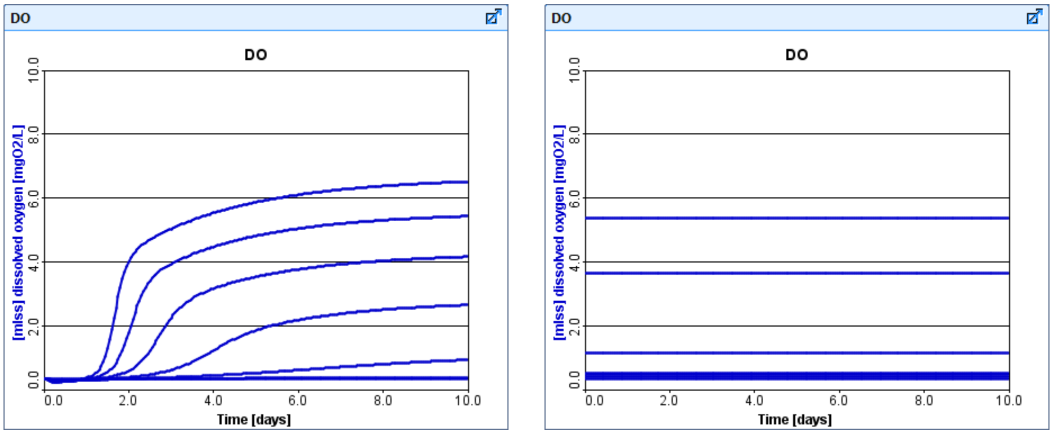
When doing a dynamic sensitivity analysis there are two important cases to consider. In the first case the initial conditions are such that the process is initially at steady state. In the second case the initial conditions are dynamic. In CHAPTER 8, these two cases were examined with regard to the interpretation of the dynamic behavior that results. In general, you should assign initial conditions carefully. The same cautions apply when performing dynamic sensitivity analyses because here too you must specify appropriate initial conditions before running simulations with different values of the independent variable.

Examples of the two types of dynamic sensitivity analysis that you can perform are shown in **Figure 9-2**.

In both cases, a dynamic simulation is run from time=0 until the **Stop Time**.

Panel A of **Figure 9-2** shows the change in dissolved oxygen (DO) concentration of the mixed liquor over time for several values of the wastage flow rate. Notice that the simulations all start at the same value, the preset initial value.

Now compare this to a similar sensitivity analysis setup shown in panel B. In panel B the simulations start at different values of DO since, in this type of analysis, the user has specified that the steady-state solver should be run before conducting the dynamic simulation.



Panel A Panel B

Figure 9-2 – Dynamic Sensitivity Analysis (Steady-State Off vs On)

There are important differences in the interpretation to be given to these two cases. In the set-up resulting in the graph in Panel A of **Figure 9-2**, the results demonstrate how the model responds to different values of the wastage rate starting from specific values of the model state variables, including the DO. In this case, it may be necessary to either extend the simulation or to perform a number of re-runs to observe the ultimate, periodic steady-state for the system. This type of analysis would be useful to answer questions such as the following:

- What is the dynamic response of the mixed liquor suspended solids concentration if the wastage rate is suddenly changed?
- How would the current DO concentration change over time if the air flow rate was suddenly turned up (or down)?
- What happens to the solids distribution over time if we quickly change to step feed operation in the plant?

In each case, there is an implied common starting point (current MLSS concentration, DO concentration or solids distribution) and a specified change in a model independent variable (wastage rate, air flow rate or step feed). The immediate, short-term response of the model is of greatest interest.

In Panel B of **Figure 9-2**, the situation is considerably different. In this case the simulation starts at a different DO value for each run. Here the steady-state solver is used to determine steady-state values for each value of the independent variable, set the initial conditions equal to these values and then started a dynamic simulation. The model is assumed to have been operating in a steady-state. At time=0 the value of the independent variable changes to that specified in the sensitivity analysis and a dynamic simulation is conducted until the specified stop time. This type of analysis is useful for answering questions such as the following:

- What variation in mixed liquor suspended solids can be expected over time for plant operation at different wastage rates?
- In the long run, how will the DO concentration vary over time for different values of the air flow rate?
- What is the solids inventory like if I use step feed all the time?

The common characteristic of these questions is the implied interest in long-term changes rather than the short-term effects of a sudden change. The initial value is not of concern; instead it is preferred to avoid the situation where the model is making a transition from one periodic steady-state to another and to examine the relative differences between these periodic steady-states.

Time Dynamic and Phase Dynamic Analyses

The results of a dynamic sensitivity analysis can be displayed in one of two ways. **Figure 9-2** shows one way, that is, selected dependent variables plotted versus time. This is referred to as a **time dynamic** sensitivity analysis.

In some cases, it is desirable to view only the end-point of the dynamic simulation for each value of the independent variable. For example, you might want to ask:

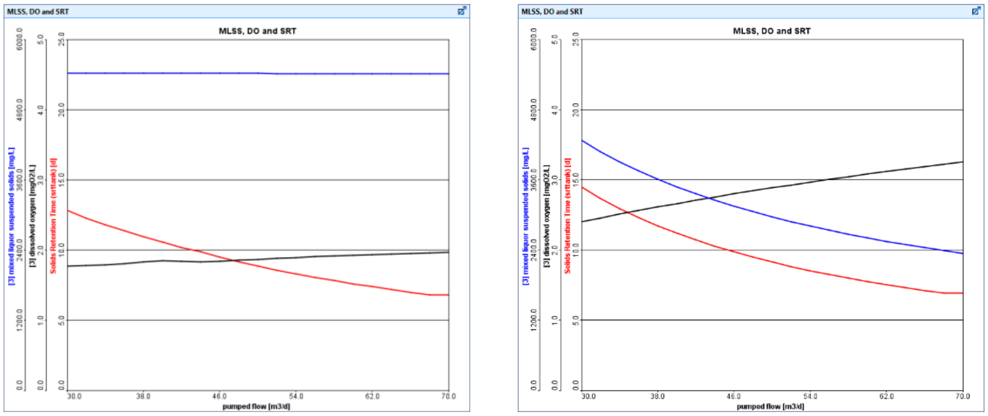
- What is the value of the mixed liquor suspended solids one and a half (1 ½) days after a sudden change in the influent flow?
- What will be the value of the DO concentration at the end of each week when the plant is subject to a different, repeating diurnal flow pattern?

The questions suggest that the simulation result of interest is the value of the dependent variable at a specific point in time. This is referred to as a phase dynamic sensitivity analysis. Finding answers to these questions requires the ability to plot the values of dependent variables at the end of a simulation against the value of the independent variable. This type of plot, referred to as a phase diagram, is shown in **Figure 9-3.**

There is one additional step to perform before initiating the analysis. A decision needs to be made whether the initial conditions are steady state or dynamic.

By selecting **Steady-State** (by checking it on the **Simulation Toolbar**) you direct GPS-X to calculate the steady-state values and use these as the initial conditions. If **Steady-State** is not selected, the manually entered initial conditions are used. These two types of analysis are referred to, respectively, as long and short-term analysis.

This difference is demonstrated in **Figure 9-3** which shows the same phase dynamic sensitivity analysis without steady-state initialization (Panel A) and with the initial conditions set using the steady-state solver (Panel B).



Panel A **Panel B**

Figure 9-3 - Phase Dynamic Sensitivity Analysis Graphs.

(Panel A) Starting from Non-Steady State.

(Panel B) Starting from Steady State

STEPS IN SENSITIVITY ANALYSIS

NOTE: Sensitivity analyses are set up after the model has been built. The procedures in the remaining sections of this chapter assume that the outputs you want to display have already been set up and that the model which is the subject of the sensitivity analyses has been built.

After creating your layout, there are a few steps to set up a sensitivity analysis (steady-state or dynamic) as described below:

1. Switch to **Simulation Mode**.

2. Specify the model parameter to serve as the independent variable (see **Creating A Control From An Independent Variable** in CHAPTER 6).

3. Enter the minimum, maximum and increment value for the independent variable and set it as the appropriate analyze controller type (see **Input Controls Properties in CHAPTER 6**).

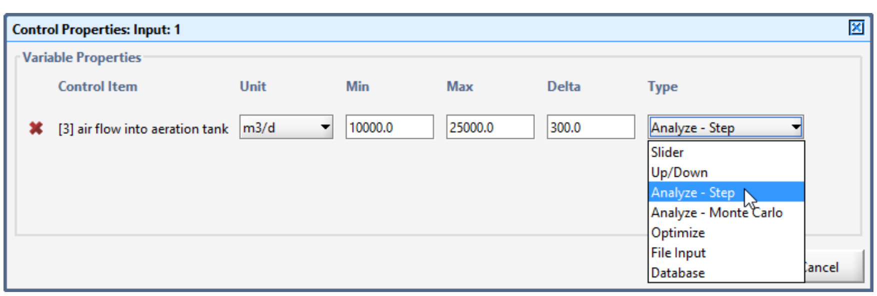


Figure 9-4 – Set Input Control to Analyze Type

4. Select the desired type of analysis (steady-state, phase dynamic, time dynamic, monte carlo) from the drop-down list accessed through the **Analyze** button on the **Main** Toolbar or the Analyze menu item in the **Tools Menu**.

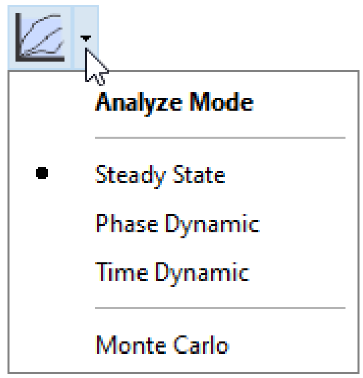


Figure 9-5 – Accessing Analyze Features

5. Turn on Analyze Mode by clicking the **Analyze** button on the **Main** Toolbar (to toggle the mode) or accessing the Analyze menu item and selecting **Analyze Mode**. The status bar at the bottom of the main window will indicate that you are in **Analyze Mode**.

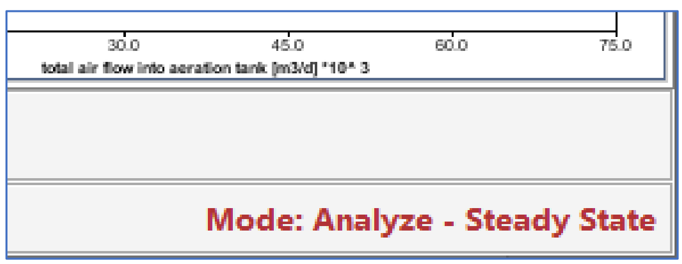


Figure 9-6 – Status Bar showing Analyze Mode

6. Select the desired Stop Time (if applicable) and start the simulation.

Important Tips

1. There can be only one independent variable so don't specify more than one variable as an analyze type control. You can set up controls of other types and they will be displayed when you open the controls window. To avoid the effects of confounding variables, the other controls should not be changed when a sensitivity analysis is being performed.

2. Once the independent variable has been specified, the control is set to a non-interactive gauge; however, the program does not change to the analyze mode until the mode is changed manually to analyze.

3. As the independent variable is incremented, the analyze controller displays its current value in a non-interactive gauge as shown in **Figure 9-7**. You can estimate the extent of completion by observing this gauge.

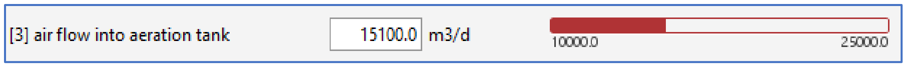


Figure 9-7 – Analyze Controller

4. The outputs that are displayed when doing a dynamic analysis will depend on whether **Time Dynamic** or **Phase Dynamic** was selected from the **Analyze** menu. For **Time Dynamic** analyses, time is plotted on the X-axis, whereas for **Phase Dynamic** analyses, the specified independent variable is plotted on the X-axis.

WHAT IS MONTE CARLO ANALYSIS?

Monte Carlo analysis may be conducted as either a steady-state or dynamic analysis. The process for setting up a steady state or dynamic Monte Carlo analysis is similar to the set-up of the dynamic and steady state analyses outlined earlier in the chapter but there are a few differences which will be covered below.

Monte Carlo analysis functions in a similar way to the step analysis method discussed earlier in this chapter; however, unlike step analysis, where the parameter is incremented by some delta value over the range, Monte Carlo analysis samples the parameter's range following a probability distribution. The results of a Monte Carlo analysis are the probabilities of particular outcomes occurring, given the parameter range and probability distribution.

Setting up Output Displays

Output from the Monte Carlo analysis can only be displayed using the **Probabilistic (Monte Carlo)** graph type.

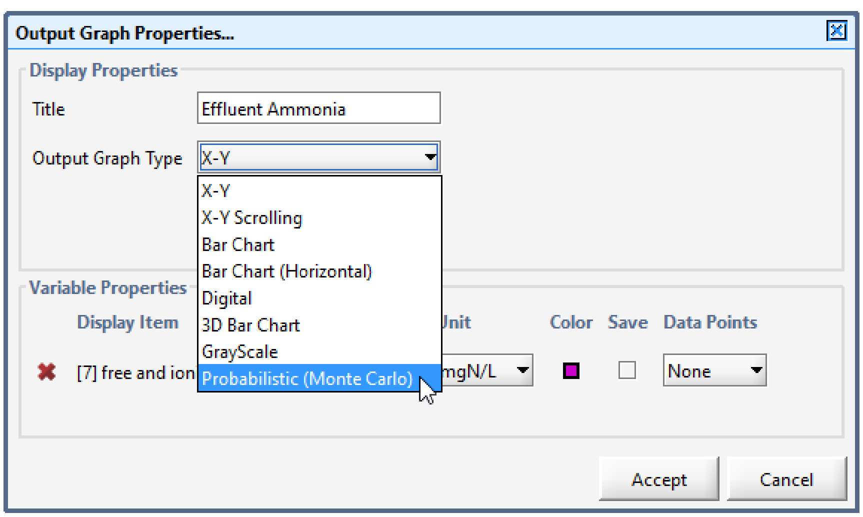


Figure 9-8 – Output Properties showing Probabilistic Option

Setting up the Input Controller and Distribution Properties

1. Specify the model parameter to serve as the independent variable (see **Creating A Control From An Independent Variable** in CHAPTER 6).

2. Enter the minimum and maximum value for the independent variable and set it as the **Analyze – Monte Carlo** controller type (see **Input Controls Properties** in CHAPTER 6).

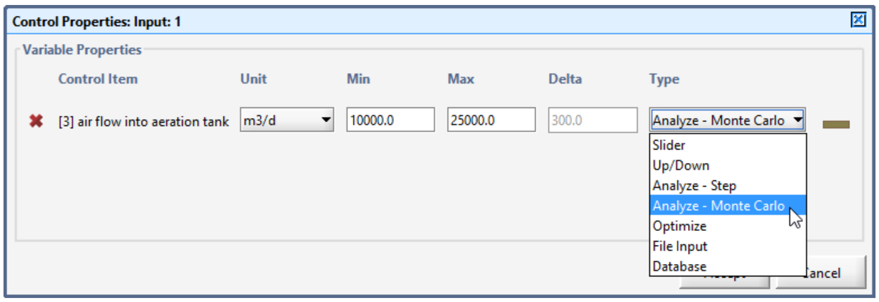


Figure 9-9 – Set Input Control to Monte Carlo Type

When **Analyze – Monte Carlo** is selected from the drop-down menu, a button will appear beside the Type column. Clicking on this button opens the ***Distribution Properties*** dialog for the Monte Carlo analysis.

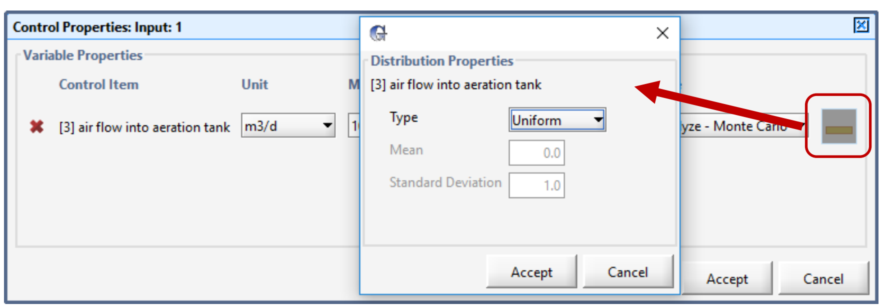


Figure 9-10 – Accessing Distribution Properties

The probability distribution may be defined as uniform, normal, or log normal. The particular normal or log normal distribution required is specified using its mean and standard deviation.

3. Select the Monte Carlo analysis from the drop-down list accessed through the **Analyze** button on the **Main** Toolbar (see **Figure 9-5**) or the Analyze menu item in the **Tools Menu.**
The status bar at the bottom of the main window will indicate that you are in **Analyze Mode**.

4. Select the desired Stop Time (if applicable) and start the simulation.

During each run, by default GPS-X selects a new set of random variables from the designated distributions. If a repeatable set of random variables is required, change the “Monte Carlo seed” value in the ***View* > *Preferences* > *Input/Output*** menu to a positive value. Setting the seed value to -1 will force GPS-X to select a new set every run.

CHAPTER 10

Optimization Tools

INTRODUCTION

The **Optimize** module is used to optimize the values of parameters from your layout. This module is an optional feature. If you have not purchased this module, contact us for pricing information.

The ***Analysis Tools*** chapter explained how to use the analyze module to study the behaviour of your models and to identify the important parameters. Once you have identified the important parameters, you can use the optimizer module to optimize the values of these parameters. Optimization involves adjusting certain model parameters to achieve a desired objective function.

The optimizer can be used to fit a model to measured data or to optimize process performance. The procedure of fitting a model to measured data is called "parameter estimation" and involves adjusting selected model parameters to achieve the best possible fit between the model responses and the measured data. Parameter estimation is an important step in preparation of a simulation model because process parameters can vary significantly from plant to plant. A model that has been fitted to actual plant data will be more useful for predicting actual plant behaviour.

Process optimization involves adjusting certain model parameters to maximize or minimize the value of a model variable or a user-defined variable. For example, you may wish to adjust certain model parameters to minimize a plant's operating cost.

The optimizer module was developed specifically to solve parameter estimation and process optimization problems involving dynamic wastewater treatment models. It can be used for both steady-state and dynamic optimization. As will be seen in this chapter, the optimizer is a valuable tool for preparing effective models of wastewater treatment facilities.

USES OF OPTIMIZATION

There are several areas where optimization can be beneficial. In wastewater treatment, the main areas of application are in model calibration (i.e. parameter estimation), process design, and process optimization.

Parameter estimation is important because once a model has been built it often becomes necessary to fit the model to real data. In fitting simulation results to real data, it is usually found that many stoichiometric parameters can be assumed constant but that physical, operational, and kinetic properties vary significantly from plant to plant. You can use the GPS-X parameter default values as a starting point, but when it is necessary to use the model for purposes of predictive simulation then calibration should be performed. This can be a simple manual or qualitative simulation in which you conduct interactive simulations or a rigorous mathematical one using formal optimization techniques.

Simulator optimization can be used for process design and plant optimization. Consider the problem of finding a plant design that meets certain effluent requirements. Another example is finding the best operating mode to reduce the loss of suspended solids during a plant upset. In both cases, there is a desired output, namely the design that meets the effluent limits and the best operating mode. If you have a model of the system, these objectives are achieved by varying plant design or operational parameters and observing the response of the model outputs.

In GPS-X, you choose the target parameters and the optimized variables of interest, and select an objective function to achieve a desired optimization. There is no limit on the number of response variables. The optimizer adjusts the selected parameters until the objective function is minimized.

ALGORITHM USED

GPS-X uses the Nelder-Mead simplex method (Press *et al*., 1986) in the optimizer module. The simplex method is a robust multi-dimensional algorithm that does not rely on gradient information. The algorithm searches through a multidimensional "surface" or space (defined by the objective function) in an organized way to find a path to a minimum value for the objective function. The procedure is sensitive to the choice of the optimization parameters. For more information on the simplex algorithm refer to the ***Optimizer*** chapter in the ***Technical Reference***.

TYPES OF OPTIMIZATION

The optimizer module is equipped to handle three different types of process measurements:

1. Time series measurements,
2. Long term operational data that are averages of the original process measurements, and
3. On-line measurements.

Each type of measurement set leads to a different type of optimization problem. The optimization problem types available in GPS-X are **Time Series** and **DPE**.

Time Series

This optimization type is the one normally used for both parameter estimation and process optimization. It is designed to handle both time series and steady-state measurements.

For parameter estimation involving a dynamic model, the data entered into the text file will be the values for each of the response variables that you would like to fit at a series of time values. The response variables are referred to as target variables. In GPS-X, the target variable can also be maximized or minimized.

Steady-State optimization is a **time series**-type optimization with only one data point for each target variable. The steady-state solver is used and the simulation has a stop time of 0.0. This type of optimization is useful for calibrating the model to data reported as daily, weekly or monthly averages. Data of this type are typically obtained from composite samples and do not accurately reflect the time dynamics of the real process. In a steady-state optimization, the average data are used as the targets and selected model parameters are adjusted to fit these targets.

GPS-X will fit your model to the measured data using the objective function that you select. On the output graphs used to display the predicted values of the model, the measured values provided in the data file will be displayed on the graphs with red diamond markers.

GPS-X will draw a new curve for the predicted values at each optimization iteration, so that you can track the progress of the optimizer. At the end of the optimization run, final predicted responses are displayed so that you can visually assess the fit. An example of this type of graph is shown in **Figure 10-1**.

You can use any of the available objective function types when doing a time series optimization. These are listed below:

1. Absolute Difference
2. Relative Difference
3. Sum of Squares
4. Relative Sum of Squares
5. Maximum Likelihood

When doing parameter estimation, the maximum likelihood or sum of squares objective functions should be used. For process design or optimization, the absolute difference objective function is the most suitable. For more detail on the different objective functions see ***Objective Function Options*** in the ***Optimizer*** chapter of the ***Technical Reference***. The optimization results you obtain can depend strongly on the chosen objective function.

Dynamic Parameter Estimation (DPE)

GPS-X has a sophisticated dynamic parameter estimation feature (DPE) designed for the estimation of time-varying parameters. It can be used with on-line data or off-line time series data. For details on using on-line data, contact Hydromantis.

The motivation behind DPE is that parameters in process models are often not constant, but vary with time. For example, the oxygen mass transfer coefficient in an aerated tank is often slowly time-varying.

Dynamic parameter estimation is useful for estimating parameters in poorly understood processes. In these cases the model structure is likely to be incorrect. As a result, the model may only be able to represent the data well over short time intervals. In this case, using DPE will help compensate for the model error and allow acceptable fitting of the measured data.

Another situation in which dynamic parameter estimation is useful is when you are interested in detecting process changes and upsets. If for example a model parameter is found to be relatively constant during normal process operation but is sensitive to process changes, you can track this parameter using the DPE feature and on-line data to help provide an early warning of process changes or upsets.

In GPS-X dynamic parameter estimation is done by applying the time series optimization approach mentioned earlier to a moving time window. Instead of estimating parameters from an entire set of data, GPS-X calculates a set of parameter estimates for each time window using the parameter estimates from the previous time window as a starting guess. This approach can be used on a data file that is continually updated with new blocks of data (online data) or on a static file of time series data. You can use any of the objective functions that are available for time series optimization when doing dynamic parameter estimation.

The length of the time window controls how often the parameters are updated. The shorter the time window, the more often the parameters are updated. When using short time windows it may be necessary to filter the data to eliminate noise so GPS-X does not fit the noise.

To ensure proper termination of the optimization routine when using the DPE feature, it is suggested that the time window and the communication interval be chosen such that the time window is an integer multiple of the communication interval.

DYNAMIC SIMULATION INITIAL CONDITIONS

When performing an optimization involving a dynamic simulation you can set the initial conditions using the steady-state solver or provide your own initial estimates of the initial conditions. This choice is dictated by the target data and knowledge about the dynamics of the system and its behavior at the start of the simulation.

If the dynamic target data include initial steady-state estimates, perhaps from time-averages, you can use the steady-state solver to establish the initial conditions for the simulation. If you know that the system is not at steady-state at the start of the simulation, you can use the initial data points in the real data set as the initial conditions. **Figure 10-1** shows an optimization of a batch process in which the initial condition represents the initial concentration of soluble substrate in a tank. The target data are shown as discrete points and simulation results for six values of the independent variable are shown as continuous curves. Steady-State initial conditions are not appropriate for this case. Since soluble substrate (a state variable) is utilized in the batch reactor, the steady-state solver would find a solution at zero substrate concentration. If the simulation started at zero initial substrate, the optimizer would not be able to fit the simulation to the target data.

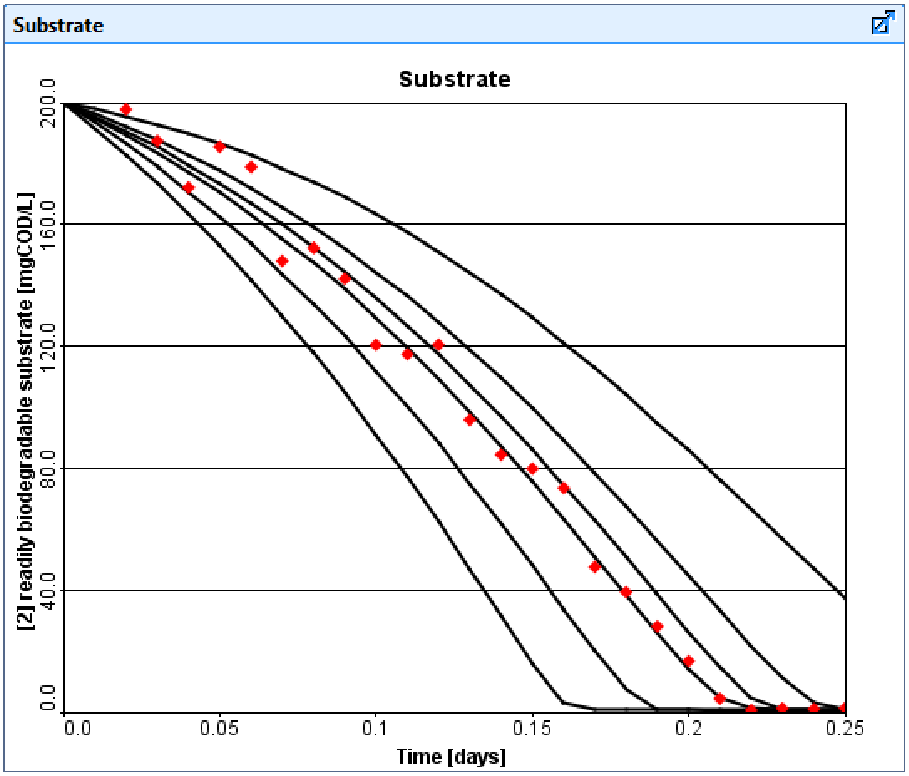


Figure 10-1 – Example of Dynamic Simulation using Initial Conditions

SELECTION OF THE OPTIMIZATION VARIABLES

Before using the optimizer to adjust parameter values it is usually best to experiment with manual adjustment of the parameters. It is a good idea to familiarize yourself with the models you prepare to understand the important relationships. By conducting interactive simulations you can observe the effects of the many model parameters on the response variables of interest. With this information, you will be able to make better judgments on appropriate independent and target variables to use in an optimization.

One common optimization problem is selection of improper optimization parameters. In setting up a manual or automatic optimization, be sure to select those parameters that have a significant effect on the specified target variables. For example, effluent ammonia concentrations are highly dependent on, and responsive to, the value of the nitrifier growth rate or reactor aeration parameters, but not so dependent on the value of the carbon-removing organism growth rate. Similarly, recycle sludge concentration is not influenced greatly by the settler model flocculation parameter whereas effluent suspended solids from the settler is influenced by this parameter. As demonstrated by these examples, selection of appropriate optimization variables requires an understanding of the structure of the model.

Including too many parameters in an optimization will result in a difficult optimization problem with a high degree of correlation between the parameters. As a result, the parameter values at the solution will have a high degree of uncertainty associated with them.

When you are satisfied that you know the important relationships and have identified appropriate target (response) variables and optimization variables, it is best to perform some manual optimizations. Set up several interactive simulations with slider controls on the optimization variables and try adjusting these as the simulation proceeds. You can plot actual data along with the target variables (see the **Variable Properties** section in CHAPTER 7) so that you can compare simulation and actual data. This type of qualitative optimization is helpful in a number of ways. For example, it helps to identify initial guesses and establish practical bounds on the optimization variables.

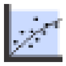
STEPS IN AN OPTIMIZATION

Optimization setup must be performed in **Modelling Mode**.

To run the optimization, you must first fully setup the target variables, optimized parameters, and objective function using the Optimizer Setup Wizard. Once the wizard is completed, the model will automatically re-build to display necessary input and output parameters. The re-build is necessary to embed optimization instructions in the simulation model.

NOTE: When doing probability optimization using input data, the data must consist of daily averages. These daily averages will be compared with the simulated daily averages calculated for the target variables. GPS-X automatically creates the daily average variable for you, based on your selected target variable. For example, if you have selected effluent TSS as a target variable, when you rebuild your layout, GPS-X will automatically generate a daily average effluent TSS variable which will be used as the target variable for the optimizer.

Optimizer Setup Wizard



The first step in setting up an optimization is to run the Optimizer Setup Wizard in **Modelling Mode**. Press the Optimize button on the **Main Toolbar**

1. **The first stage is selecting the desired target variables**. Navigate through the variable tree to find and select the desired variable.

   The target variables are the variables that you would like to minimize, maximize, or fit to the measured data. The target variables are often the same variables as those you would plot in a normal-mode simulation.

   For time series and DPE optimizations you can select other target variables in this way but keep in mind that data requirements and calculation times required for multi-target optimizations can become prohibitive.

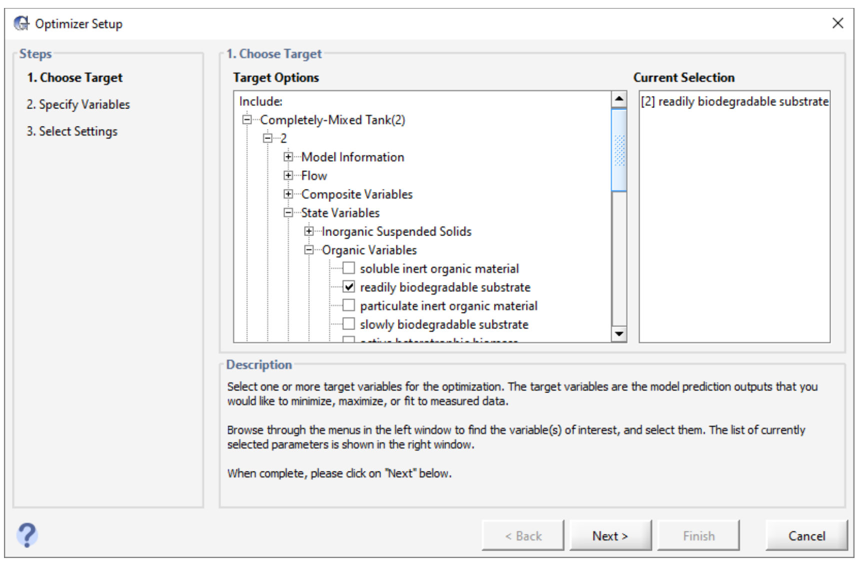


Figure 10-2 – Optimizer Setup Wizard (Specify Target Variables)

   Click "Next" to move on to the next stage.

2. **Select the desired optimized variables** by ticking the appropriate boxes. The optimization variables will be adjusted by the optimizer as it attempts to minimize the objective function.

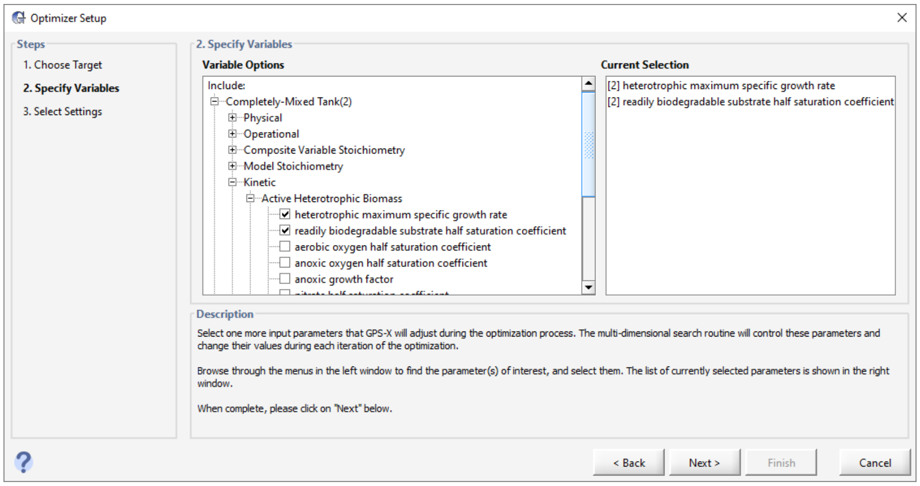


Figure 10-3 – Optimizer Setup Wizard (Specify Optimize Variables)

Click "Next" to move on to the next stage.

3. **Specify the desired target comparison, objective function and optimization type**. The target variables can be minimized, maximized, or fit to an input data. The input data can be imported into GPS-X by selecting **Fit to Data** under Target Comparison and clicking on the **Add Data File** button.

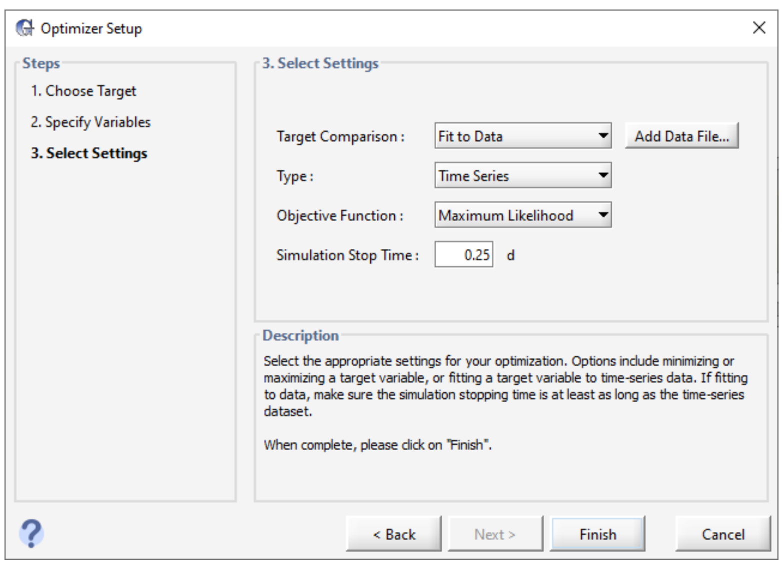


Figure 10-4 – Optimizer Setup Wizard (Specify Optimizer Settings)

4. **Press 'Finish'**. Once you press finish, the model will automatically switch to simulation mode and re-build. Notice that the layout is now in Mode Optimizer-Time Series. A new input tab labeled Optimization Parameters and a new output graph tab labeled Optimization Results will be automatically generated. The Optimization Parameters input tab contains the optimized variables and the

Optimization Results output tab contains the graphs of target variables the user specified.

If you load a layout where the Optimizer Setup Wizard has already been run, the simulation will be in ‘normal’ mode. To turn on the optimizer, press the Optimize button on the **Main** Toolbar.

The optimization type and objective function can also be changed in **Simulation Mode** instead of re-running the wizard. For details on how to do that, see the description of the **Optimize** menu in CHAPTER 1.

Running the Optimizer

To conduct an optimization, you must be in **Simulation Mode**, and you must turn on the optimizer (if it isn’t automatically turned on after going through the Optimizer Setup Wizard described above).

To run an optimization, complete the following steps:

1. Be sure you are in **Simulation Mode**.

2. Click on the **Optimize** button to switch to **Optimize Mode**. Alternatively, you can right-click on the inverted triangle beside the optimize icon, and then select **Optimize Mode**. If you have not selected the **Type** and the **Objective Function**, do this now. A message indicating you are in **Optimize Mode** will be displayed in the status bar.

3. Set the **Stop Time** and the **Communication Interval** to the desired values.

4. Click on the **Start** button on the **Simulation Toolbar**.

As the optimization proceeds, on the output graph several different simulation curves will be overlaid and these curves should begin to converge to the real target data.

The number of curves shown on the graph can be set by going to ***View > Preferences*** and on the “Input/Output” tab setting the **Number of runs displayed (analyze/optimize)** parameter.

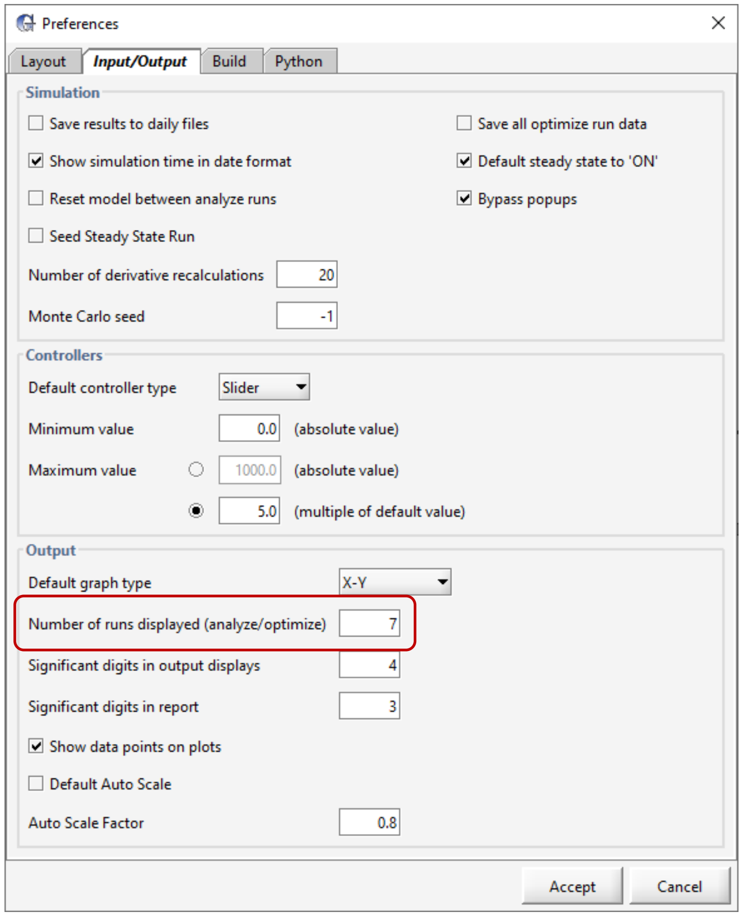


Figure 10-5 – Preferences (Number of runs displayed)

You can watch the input controllers to get a feel for the way the optimizer makes changes to the parameters as it attempts to match the simulation results to the target data.

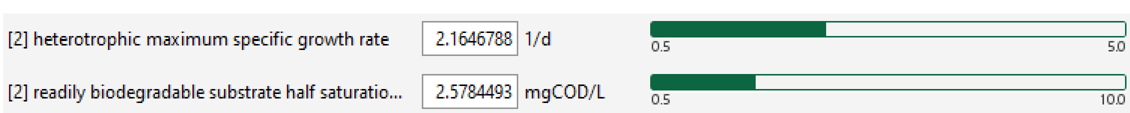


Figure 10-6 – Optimization Parameters as Input Controllers

Additional optimization results are displayed in the **Command Window**. These include the iteration report, the final solution, and a number of statistics that allow you to assess how well the model fits the data (see the **Command Window** section in CHAPTER 8 for information on how to view and use this feature). See ***Appendix B: The Optimizer Solution Report*** in the Optimizer chapter of the ***Technical Reference*** for a description of the information that is printed to the **Command Window**.

The optimization routine can take anywhere from a few seconds to a few hours to complete depending on the number of independent variables, model complexity, termination criteria, etc. It is best to start with a single parameter optimization and a small amount of data until you are accustomed to the speed and resource requirements for your model and machine and the appropriate optimization parameters to use.

ADVANCED OPTIMIZER SETTINGS

In most cases, the default optimizer settings will be sufficient but more advanced users by need to change some of the values. These settings can be located in the ***Layout* > *General Data*** menus and details about those settings can be found in the Optimizer chapter of the ***Technical Reference.***

TROUBLESHOOTING

Here are some guidelines for correcting common problems that occur:

- Make sure that you have changed to the appropriate optimize mode. You can verify by checking the main window status bar (lower right corner).

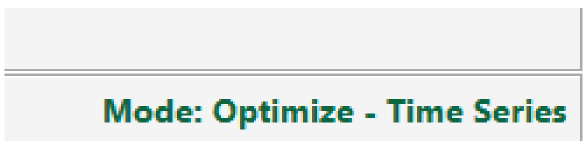


Figure 10-7 – Status Bar showing Optimize Mode

- Check to make sure the target data file has been set-up correctly. If the data file has been read by GPS-X, then the file name will be displayed in the Command Window. File input methods (via text files or Excel spreadsheets) are described in the **Using File Input Controllers** section of CHAPTER 6 .

- Use the variables that have a strong effect on the target or dependent variables. If the optimization fails to minimize the objective function (there is a poor fit between target data and simulation results), it may be due to improper selection of the optimization variables.

- The simplex method settings can have a strong effect on the optimization. Read the ***Optimizer Description*** in the Optimizer chapter of the ***Technical Reference*** for a description of these settings.

- Check that the optimizer converges to a reasonable solution. There may be many local minima, and possible more than one global minimum. It may be necessary to conduct several runs from different starting guesses to find the lowest minimum.

CHAPTER 11

Customizing GPS-X

INTRODUCTION

The models in GPS-X are varied and the most comprehensive available in a wastewater treatment process simulator. In developing a model of your plant, you may require special modifications or additions to GPS-X. For example, you may want to add your own calculated variables or enter new equations. GPS-X is designed to be flexible and provides several ways to handle these special situations. In most cases you will find that the existing models provide a good foundation for customization and that with minor changes you can easily tailor the model to suit your needs. When making major changes to the process models or developing new models, it is recommended that you use the Model Developer tool that is available with GPS-X (contact Hydromantis for details).

This chapter covers the methods used to make modifications to the forms used in GPS-X and to add new variables and equations. The material includes an explanation of the general structure of the GPS-X software system and then outlines the various ways in which you can customize the software to suit your needs.

In general, this chapter assumes a more advanced level of understanding, both of the GPS-X system itself and other aspects of dynamic modelling and simulation. The material presented in this chapter is supported by examples rather than lists of specific procedures as in earlier chapters. The examples will help to reinforce the material and give you a more solid foundation for making your own modifications.

To complete the examples, it is helpful if you have an understanding of the following:

- Advanced Continuous Simulation Language (ACSL™)
- FORTRAN programming language
- Dynamic modelling of the unit processes in a wastewater treatment facility

ACSL is the simulation sub-system used by GPS-X. Some model modifications require an understanding of the ways in which ACSL is used to implement the model. A complete examination of the features and capabilities of ACSL is provided in the ***ACSL Reference Manual*** (Contact Hydromantis for details).

Because the general programming language used by GPS-X and ACSL is FORTRAN, knowledge of the programming constructs in this computer language and tools for debugging FORTRAN programs may be required. Finally, since you may be making additions to the unit process models, knowledge of the fundamentals of dynamic modelling and simulation of wastewater treatment processes is important.

If you are unfamiliar with one or more of these areas, you may still be able to make the necessary modifications by following closely the examples in this chapter.

TYPES OF CUSTOMIZATION

There are two approaches to customization in GPS-X. The first approach involves making modifications to a specific GPS-X layout. In this case, the changes only apply to the layout itself and do not affect newly created layouts. The second approach is to create a custom library containing the changes. In this case, all layouts created using this library will make use of the modifications.

The first approach is useful if you want to add new variables or equations to a specific layout and you do not need to re-use the modifications in other layouts. For example, you may wish to create a new composite variable that is a variation on one of the existing composite variables. Methods for making the changes associated with the first approach are presented in the **Customizing Layouts** section of this chapter. If you inadvertently delete of corrupt these files, GPS-X can always be re-installed. If you inadvertently delete or corrupt these files, GPS-X can always be re-installed.

Customizing Layouts

The second approach is useful if you would like to change the variable names used in the GPS-X forms or change the default values used in the forms and then re-use these modifications in a number of layouts. The second approach involves modifications to model libraries, and is a procedure that requires some understanding of the structure of the GPS-X software system. Procedures for making these kinds of changes are discussed in **Customizing Libraries** section in this chapter.

GPS-X SOFTWARE SYSTEM

File System

The GPS-X software consists of three types of files that are stored in appropriate directories on your machine:

1. Program executable files
2. Model library and auxiliary files
3. Layout files

Program Executable Files

The program executable files are binary files that cannot be altered. They include the GPS-X executable, the ACSL executable and other secondary binary files used by GPS-X while it is running. The GPS-X executable files are stored in the `\bin` sub-directory in the GPS-X installation directory. The ACSL executable files are stored in the `\acsl11` sub-directory in the GPS-X installation directory.

The layouts that you create are executable files; however, they are stored in the directories in which you create them.

Model Library and Auxiliary Files

The GPS-X model libraries are stored in sub-directories in the GPS-X installation directory (eg. the *cnlib* directory). Each of these libraries contains a binary file, `gpsxm.bin` that contains the lower level ACSL macros that are used to construct the process object models. This file also contains extensions to the ACSL operator set developed specifically for GPS-X. This file cannot be modified.

The GPS-X libraries contain templates for the layout files (discussed in the next section), and model auxiliary files that are specific to the unit process models built into the GPS-X software. The auxiliary files have the following general form:

- `modelnme`**`.cfg`** (model file)
- `modelcfg`**`.con`**
- `modelcfg`**`.ini`**
- `modelcfg`**`.dis`**

Where: **`cfg = asp, pft, pfm, pmp`**

Here, `modelname` (eight alphanumeric characters maximum), and `model` are identifiers for each model type. The suffixes `asp, pft` etc., refer to the object type. The name for the `.con, .ini,` and `.dis` files are constructed by taking the first five characters of `modelname` and appending the 3-letter object type specification. (`asp, pft`, etc.). A complete list of these names is given in the ***Technical Reference***.

The files listed above are library-dependent, so the format of the files depends on which GPS-X model library you are using (i.e. CN, CNP, Mantislib, etc.)

The `modelname.cfg` files contain the defining ACSL macros for the process models in GPS-X. These macros call the lower level macros found in `gpsxm.bin`.

The `modelcfg.con, modelcfg.ini` and `modelcfg.dis` files are used to define the set-up of the forms accessed through each process object's process data menu. The `.con` files are used to define the forms containing the parameters and constants. The `.ini` files are used to define the initialization forms while the `.dis` files are used to define the display variable forms.

Model auxiliary files are text (ASCII) files, and can be edited with any text editor. By making changes to these files, you can modify initialization parameters and model constants and change the look of the menus that are displayed, and the text in those forms. You can also modify the model code, if necessary (contact Hydromantis for details). These types of changes involve changing the model library and will apply to all layouts created using this library (see the **Customizing Libraries** section in this chapter for details)

Layout Files

Layout files are specific to the layouts you prepare on the GPS-X drawing board. These include:

- `layoutname.usr`
- `layoutname.var`
- `layoutname.con`
- `layoutname.ini`
- `layoutname.cmd`

Where: `layoutname` is the file name you specify when you **Save** a GPS-X layout.

Layout files are used to make changes to a specific layout. You can create new variables and equations, and define your own custom data entry and display variable forms. The `.usr` (user) file is used to enter modelling equations. The `.con`, `.ini`, and `.var` files are used to define custom forms for the new variables.

Data Entry and Display Variable Forms

In GPS-X, data entry and display variable forms, such as those shown in **Figure 11-1** are created dynamically when you make a selection from the process data pop-up menu.

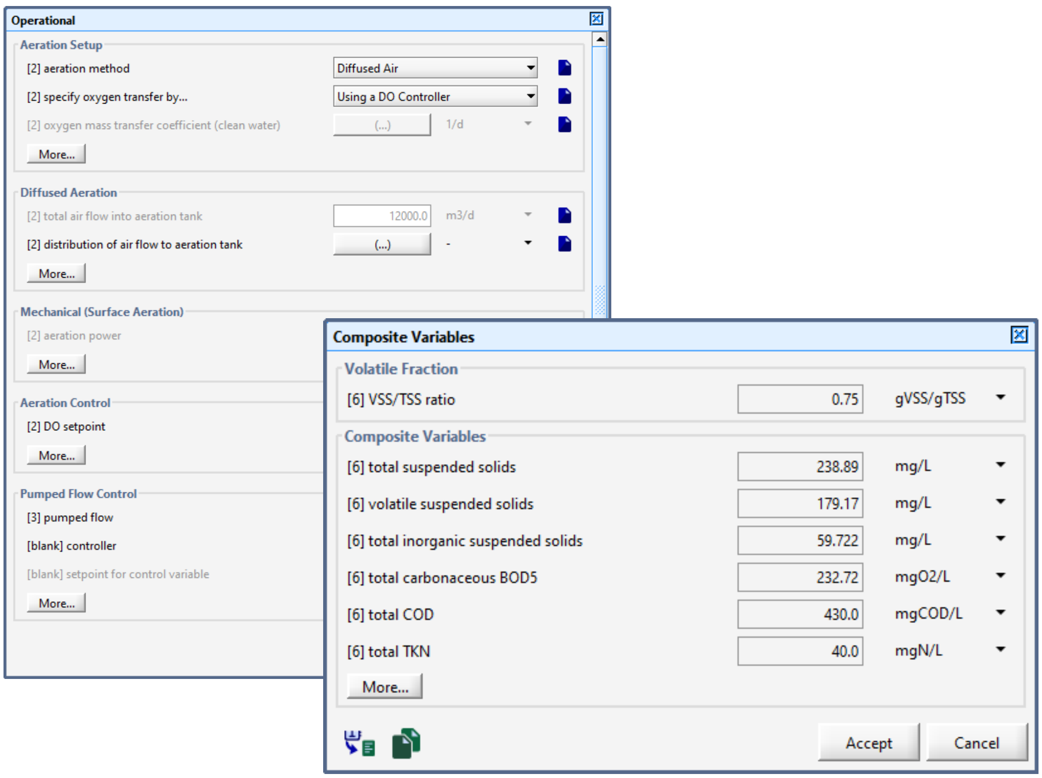


Figure 11-1 – Data Entry (left) and Display Variable (right) Forms

The numerical constants and text strings used in these forms are read from the model `.con`, `.ini`, and `.var` files. This makes it easy to customize GPS-X to display alternative labels in the data entry and display variable forms or to add entries in these forms.

Table 11-1 shows which file to edit for the different GPS-X forms. When you make changes to these files in a GPS-X library, the changes apply to all newly created layouts that use that library. Therefore, it is a good idea to make a copy of the library you are modifying and make changes to this new library so that the original library is left unchanged.

Table 11-1 - Files to Edit for the Diffrent Forms

To modify entries for:	Edit the file with extension:
model parameters or constants	`.con`
initial conditions	`.ini`
display variables	`.var`

Model Constants and Initialization Menus and Forms

Changes to the values of model constants and initial conditions displayed in the entry forms can be made by editing the associated `.con` and `.ini` files. The first few lines of the CN library model constants file `mantipft.con`, which can be found in the `\cnlib` sub-directory, are shown in **Figure 11-2**.

```
!MENU ITEM:     !Physical
!HEADER:        !Dimensions
      parameter(n&o      = 4)         ! number of reactors                !
      constant h&o = 4                   !tank depth                           !m
       xstring vsetup&o = 1          !volume setup method :1 Volume Fractions/2 Individual Volumes/ !
!HEADER:!Individual Volumes
      constant vlcon&o    = 250,250,250,250 ! individual volumes      !m3   ;vsetup&o=2
!HEADER:!Volume Fractions
      constant vmcon&o    =1000.0     ! maximum volume                    !m3 ;vsetup&o=1
      constant fvcon&o    =0.25,0.25,0.25,0.25 !volume fractions !-        ;vsetup&o=1
!MORE
```

Figure 11-2 – Example of mantipft.con File

When you display the data entry form for the physical parameters of the plug-flow tank, the text displayed for the tank volume is shown in this file as `maximum volume`. Similarly, the units are displayed as `m3`.

Some parameters are "greyed-out" (made inactive) depending on the settings of other parameters in the menu. To make a parameter become inactive, include a semi-colon at the end of the line, followed by the parameter setting that would make the parameter active. The absence of a semi-colon at the end of the line is interpreted as meaning that the parameter should never be "greyed-out".

If you want to change the displayed text to `maximum liquid volume`, then you must edit this file and enter in the new text string. This modification is shown in **Figure 11-3**.

```
!MENU ITEM:     !Physical
!HEADER:        !Dimensions
      parameter(n&o      = 4)         ! number of reactors                !
      constant h&o = 4                   !tank depth                           !m
       xstring vsetup&o = 1          !volume setup method :1 Volume Fractions/2 Individual Volumes/ !
!HEADER:!Individual Volumes
      constant vlcon&o    = 250,250,250,250 ! individual volumes      !m3   ;vsetup&o=2
!HEADER:!Volume Fractions
      constant vmcon&o    =1000.0     ! maximum liquid volume                  !m3 ;vsetup&o=1
      constant fvcon&o    =0.25,0.25,0.25,0.25 !volume fractions !-        ;vsetup&o=1
!MORE
```

Figure 11-3 – Modified mantipft.con File

DO NOT change the units. The algorithm calculations where these variables are used assume certain units (ie. they are the preset units displayed in these files).

The flags `!MORE` and `!NOMORE` are used to define which variables are moved to the secondary data forms accessed through the **More…** buttons.

If you make changes of this type to `.con`, `.ini`, or `.var` files in the library subdirectories, you must re-start GPS-X in order for the changes to take effect. GPS-X reads this information once during program initialization. To preserve the original files in a library directory, you should make a back-up of the files you intend to change. If you inadvertently delete or corrupt these files, GPS-X can always be re-installed.

Display Variable Menus and Forms

Display variable menus and forms are also customizable. A typical display variable menu is shown in **Figure 11-4**. The text that is displayed in the **Output Variables** sub-menu can be changed by making modifications to the associated `.dis` file for that object and library.

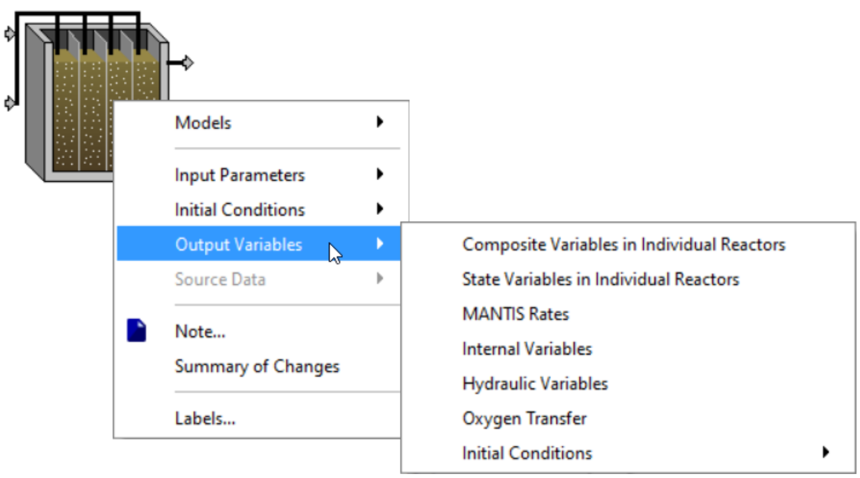


Figure 11-4 – Output Variables in Process Menu

```
in!modelid!flow!composite!state!stoichiometry!
rin!modelid!flow!composite!state!stoichiometry!
o!modelid!flow!compmlss!state!stoichiometry!reactor!cfstr!loadingras!sost!sotr!qairsum!totalpower!aer2cont!airpumpmisc!con!
p!modelid!flow!compmlss!state!stoichiometry!pfcont!con!
l!compmlssvector!statevector!mantisint!cfstr!aer!ini!
```

Figure 11-5 – Output Variables File (mantipft.dis)

Figure 11-5 shows the `.dis` file which is used to build the menu shown in **Figure 11-4.**

A `.dis` file contains a line for each display variable point in an object. Each display variable point is associated with a physical location or flow stream on the real object being modelled. Model variables are defined for each of these points because, in general, the variable values are different.

For example, plug flow tank influent values are different from the effluent values. Each display variable point is signified by a special identifier. These are as follows:

- `in` – input flow stream for all objects, except pfm
- `i1` – first input flow stream of pfm object
- `i2` – second input flow stream of pfm object
- `rin` – recycle flow stream
- `o` – output flow stream
- `p` – pumped flow stream
- `s` – underflow stream
- `g` – gas flow stream from digester objects (dig)
- `l` – internal

The identifiers are listed first on each line of the .dis file to indicate that what follows are a list of `.var` files (ie. the filename excluding the `.var` extension) contained in the same directory that should be used to construct the **Output Variables** sub-menu at that display variable point.

For example, the last line of **Figure 11-5** is used to construct the menu shown **Figure 11-4**. Each of the six identifiers separated by exclamation points are used to construct the name of a `.var` file where GPS-X gets information on how to construct additional sub-menus (if desired) and the display variable forms.

In this case, the files are:

1. compmlssvector.var
2. statevector.var
3. mantisint.var
4. cfstr.var
5. aer.var
6. ini.var

The format for a typical `.var` file is shown in below.

```
!MENU ITEM:       !Composite Variables in Individual Reactors
!HEADER: !Volatile Fraction
display ivt         !VSS/TSS ratio  in individual reactors!gVSS/gTSS
!HEADER: !Composite Variables
display x         !mixed liquor suspended solids in individual reactors!g/m3
display vss       !mixed liquor volatile suspended solids in individual reactors!g/m3
display xiss      !total inorganic suspended solids in individual reactors!g/m3
display bod       !total carbonaceous BOD5 in individual reactors!gO2/m3
display cod       !total COD in individual reactors!gCOD/m3
display tkn       !total TKN in individual reactors!gN/m3
!MORE
!HEADER:          !Additional Composite Variables
display sbod!filtered carbonaceous BOD5 in individual reactors!gO2/m3
display xbod!particulate carbonaceous BOD5 in individual reactors!gO2/m3
display sbodu!filtered ultimate carbonaceous BOD in individual reactors!gO2/m3
display xbodu!particulate ultimate carbonaceous BOD in individual reactors!gO2/m3
display bodu!total ultimate carbonaceous BOD in individual reactors!gO2/m3
display scod!filtered COD in individual reactors!gCOD/m3
display xcod!particulate COD in individual reactors!gCOD/m3
display stkn!filtered TKN in individual reactors!gN/m3
display xtkn!particulate TKN in individual reactors!gN/m3
display tn!total nitrogen in individual reactors!gN/m3
!NOMORE
```

Figure 11-6 – Output Variables Submenu File (compmlssvector.var)

By making changes to the .dis and .var files in the appropriate model library you can customize and add to the text and variable values displayed in GPS-X display variable forms. Examine the format of the existing files to gain an understanding of the structure of these forms. You can make changes to the original files, but it is best to first backup these files or copy the library files to a different directory.[12] If you inadvertently delete or corrupt these files, GPS-X can always be re-installed.

CUSTOMIZING LAYOUTS

For every model prepared in GPS-X a special file, called `layoutname.usr` file, is provided for entering code. This code is inserted into the model during the build process. This provides a mechanism for defining new equations, secondary variables, modifying the initialization process and even extensive code additions to add new capabilities such as an automatic controller.

The `.usr` file is created for every model constructed in GPS-X and is stored in the same directory as the model layout.

A typical `.usr` file is shown in **Figure 11-7**. The file is divided into five different sections, corresponding to the four explicit structure sections of the ACSL simulation language[13], plus an additional section for entering ACSL macro definitions. You can enter FORTRAN-language statements in these sections to define new variables using existing variables as building blocks.

[12] If you decide to create a new library directory and want to use this library with GPS-X, you must select ***View > Preferences >Layout***, and then choose the appropriate library from the **Default Library** drop-down menu.
[13] See ***Chapter 3, Program Structure* of the *ACSL Reference Manual*** for more information on the explicit program structure in ACSL.

```
!GPS-X Version 8.0 User File
!*****************************************************************
!PUT USER DEFINED MACROS HERE
!This section is for macro definitions

!*****************************************************************
macro userinitialsection
!INITIAL SECTION
!Macros called here will be executed in the initial section
!Don't put macro definitions here

macro end
!*****************************************************************
macro userderivativesection
!DERIVATIVE SECTION
!Macros called here will be executed in the derivative section
!Don't put macro definitions here

macro end
!*****************************************************************
macro userdynamicsection
!DYNAMIC AND DISCRETE SECTIONS
!Macros called here will be executed in the dynamic section.
!(discretes plus code to be executed every communication interval)
!Don't put macro definitions here

macro end
!*****************************************************************
macro userterminalsection
!TERMINAL SECTION
!Macros called here will be executed in the terminal section
!Don't put macro definitions here

macro end
!*****************************************************************
```

Figure 11-7 – Empty .usr File

User Defined Macros

The user defined macros section is used to define your own ACSL macros that can be called in other sections of the user file.

Initial Section

The initial section is used for calculations that only need to be performed once before the dynamic simulation process starts. Any variables that do not change during a simulation can be calculated in the initial section.

Derivative Section

The derivative section is where you place your modelling equations (or calls to macros that set up the modelling equations). You can define differential equations here and have ACSL integrate them. Also, you can place algebraic equations in this section and have ACSL solve them. The derivative section is executed at every integration step taken by ACSL.

Dynamic and Discrete Section

The dynamic and discrete section is used for defining discrete events that happen at certain intervals and for performing output related calculations (e.g. unit conversions). For example, you can define a discrete section using the ACSL keyword, discrete, that executes code at a certain defined interval or according to a schedule.

You can perform calculations that only need to be performed at every communication interval. The communication interval is the interval at which data is transferred between ACSL and GPS-X for the purposes of recording results for output. Any statements placed in the dynamic and discrete section that are not enclosed within an ACSL discrete statement are executed at every communication interval.

Terminal Section

The terminal section is for calculations that only need to be performed at the end of the simulation.

In addition to ACSL statements, you can enter FORTRAN statements in the user file.

The `.usr` file provides an easy way to extend the simulation capabilities of GPS-X. The GPS-X user-interface is designed to be a generic front-end to the simulator module. Currently, GPS-X is designed to address unit process objects and models in wastewater treatment; however, it is possible to extend the process object table and model library to include other types of unit process objects and models, for example, those used in the water treatment field.

The next example looks at how a simple model can be added to the layout using these customization features.

Adding a Simple Model to a Layout – Example

Adding new models can be accomplished by entering the necessary code in the `layoutname.usr`, `layoutname.con`, `layoutname.ini`, and `layoutname.var` files.

For example, assume you wanted to model a simple first-order decay process such as:

$$\frac{dC}{dt} = rate(C)$$

A decay constant must first be defined so that you can set its value in the GPS-X interface. An initial condition for variable C must be specified. To define these variables, and their default values, edit the `layoutname.con` file so that it looks like the figure below.

```
!Follow syntax of library .con files!
!MENU ITEM:!User Defined Constants
!HEADER:!User Defined Constants
constant rate = -10 !decay rate               !1/d
constant ic = 100   !initial concentration    !g/m3
```

Figure 11-8 – Example of a Simple layoutname.con File

If you want to make a more complex `.con` file, copy an existing `.con` file from one of the libraries located in the ***GPS-X installation directory***, and use it as a template.

Next, enter the rate expression by adding the following lines in the **Derivative Section** of the `layoutname.usr` file:

```
dc = rate*c
c = integ (dc, ic)
```

The order in which you enter these lines of code is not important since the translator module automatically re-orders the equations as appropriate.

The completed `.usr` file should look like the figure below.

```
!GPS-X Version 8.0 User File
!**********************************************************************
!PUT USER DEFINED MACROS HERE
!This section is for macro definitions

!**********************************************************************
macro userinitialsection
!INITIAL SECTION
!Macros called here will be executed in the initial section
!Don't put macro definitions here

macro end
!**********************************************************************
macro userderivativesection
!DERIVATIVE SECTION
!Macros called here will be executed in the derivative section
!Don't put macro definitions here

dc = rate*c
c = integ(dc, ic)

macro end
!**********************************************************************
macro userdynamicsection
!DYNAMIC AND DISCRETE SECTIONS
!Macros called here will be executed in the dynamic section.
!(discretes plus code to be executed every communication interval)
!Don't put macro definitions here

macro end
!**********************************************************************
macro userterminalsection
!TERMINAL SECTION
!Macros called here will be executed in the terminal section
!Don't put macro definitions here

macro end
!**********************************************************************
```

Figure 11-9 – Example of a Simple layoutname.user File

To be able to display the new variable, `C`, enter the following line in the `layoutname.var` file:

```
display c !concentration !g/m3
```

The completed `.var` file should look like the figure below.

```
!Follow syntax of library .var files!
!MENU ITEM:!User Defined Output Variables
!HEADER:!
display c   !concentration  !g/m3
```

Figure 11-10 – Example of a Simple layoutname.var File

If you want to make a more complex `.var` file, copy an existing `.var` file from one of the libraries located in the ***GPS-X installation directory*** and use it as a template.

Once these changes are made, you must re-build the model.

When you run the model, the simulator module will evaluate the equations you entered into the **Derivative Section** at each integration time step.

You can set up output displays using the variable `c` and, since it is a valid model variable, you can use it in the evaluation of secondary variables.

The variables, including constants, initial conditions and those for display, that were setup in the `.con`, `.ini`, and `.var` files can be accessed in the User section of the **General Data** menu.

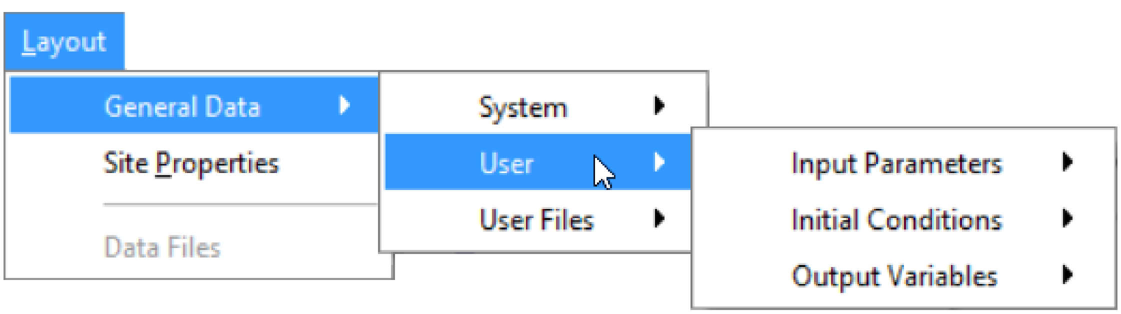


Figure 11-11 – Accessing the User Menu

The **Input Parameters** item allows access to the data entry form containing the variables entered in the .con file.

The **Initial Conditions** item allows access to the data entry form containing the variables entered in the `.ini` file.

And the **Output Variables** item allows access to the form containing the variables entered in the `.var` file.

Customizing Libraries

When you make modifications to GPS-X libraries, they apply to any subsequent layout prepared using the modified library. This is significantly different from the customizations described in the previous sections, which dealt with modifications to a specific layout only.

Library customization is used when you want to make re-usable changes to the model auxiliary files.

You can also make changes to the model code but this requires a detailed knowledge of the lower-level macros used in GPS-X. If you wish to modify the model code, contact Hydromantis for assistance.

To successfully modify an existing GPS-X library, it is important to have some familiarity with the models in the library and in general the common basis on which GPS-X organizes these libraries. Before starting this type of customization, it is a good idea to read the material on libraries in the ***Technical Reference***.

The steps to complete a library customization are listed below:

1. Choose the library to modify and copy the necessary files to a different directory.
2. Modify the appropriate model auxiliary files (i.e. the `.ini, .con`, `.dis`, and `.var` files)
3. Start GPS-X and re-build the model.

Modify the auxiliary files as discussed in **Model Constants and Initialization Menus and Forms**, and **Display Variable Menus and Forms** sections in this chapter.

Once the changes have been made, you can start GPS-X with the new modified library and build the model from the GPS-X interface. To use the new library select ***View > Preferences > Layout*** and then choose the appropriate library from the **Default Library** drop-down menu. GPS-X will automatically find all libraries in the **GPS-X installation directory**.

TROUBLESHOOTING

Here are some guidelines to correct common problems that occur in model customization:

- Errors in the model library can cause GPS-X to terminate upon loading. If, after creating a new model, GPS-X does not start normally, carefully check that all appropriate model files exist and that they have the proper format. If you discover errors of this type, make the necessary changes and reload GPS-X. If you continue to experience problems, move the new files to another directory and try again. If problems persist, they are due to corrupt files in your library directory. Either re-copy all the library files from the appropriate library directory or if you are working within the original library directory, reinstall GPS-X from the original distribution media.

- During compilation of a layout with new user defined code, any ACSL error messages will be reported in the Building Model window (see **Figure 8-12**). You will not be able to run a model until these compilation errors are corrected. See the ***ACSL Reference Manual, Appendix F*** for a list of the ACSL error messages.

  Common compilation errors include the following:

  Undefined variables

  Check that all of the new variables have been correctly declared in your model file and that any variable that is an input to the model is defined and assigned a value in the `.con` file

  Syntax errors

  The Building Model window may display the location of the error. In any case, check for typing mistakes and for lines which may exceed the 72 characters maximum.

  Re-computing a variable defined as a constant

  This means that one of the variables defined in the .con file has been placed on the left hand side of an equal sign somewhere in the model file.

- Run-time errors may be reported (when the model is loaded) in the **Command Window**, including immediate termination of the model load process. These are due to ACSL run-time faults. Common causes for ACSL run-time problems include the following: Uninitialized variables, Division by zero.

  If the model remains loaded when these types of errors occur, try displaying the values of any new variables that you have created. In the Command line of the **Command Window**, issue the display command by typing ‘display x2’, where x2 is the name of any new variable you have defined. Press return to issue the command. The variable and its value should be displayed in the **Command Window**. Repeat this process until you discover a variable which is uninitialized (equal to 5.55e 33), or equal to zero (as division by zero may be the problem). You can correct this fault by modifying the uninitialized variable in the `layoutname.con` file or the `layoutname.ini` file. If the model does not load, then it will be necessary to de-bug the model independent of the GPS-X program.

CHAPTER 12

Units Conversion

INTRODUCTION

When using GPS-X, a better understanding of the plant can be gained if the units are presented to the user in a form that is consistent with data available at the plant. This is useful for preventing data conversion errors, and for allowing the direct use of measured plant data, regardless of the units used for measurement. To facilitate this, unit conversion is provided in GPS-X. This allows the user to select which unit will be used to present a particular variable or parameter.

UNIT SYSTEMS

Two systems of units are provided with GPS-X, an SI system and a US system. These systems of units define the default set of units that appear in all menus, input controls, and displays.

Current Layout

The desired unit system is saved for each individual layout. The current system for the layout is displayed (and can be changed) via the Units drop-down menu on the **Main** Toolbar as shown below.

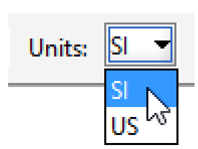


Figure 12-1 –Unit System for the Current Layout

Default for New Layouts

The default system that newly create layouts will use is selected in the ***View > Preferences*** dialog on the 'Layout' tab.

SELECTING INDIVIDUAL UNITS

Almost everywhere a unit is displayed (both inputs and outputs), you click left or right click on it to change it to a different compatible unit.

UNITS DATA FILE

All the unit conversion information in GPS-X is in a data file called `"units.cvt"`. There are two main parts to this file.

The first section is used to define the default system of units. The file provided with GPS-X provides a list of SI units and US units**. The SI units listed in this first section of the units file must not be modified under any circumstances**. The SI unit list must not be modified since it corresponds to all the internal GPS-X units. All changes in units are based on conversions from the default SI units to the unit of choice.

The second section of `"units.cvt"` contains conversion factors to convert from the SI units to any comparable unit. The default list is quite comprehensive, but the user may add other unit conversions if desired.

The file `"units.cvt"` is located in the */bin/gpsx/resources* subdirectory of the GPS-X installation directory.

Customizing a Unit System

In the units files, the first section begins with the header:

```
!SYSTEMS: SI US
```

On each subsequent line in this section of the supplied data file, the first unit is the default SI unit. The second unit is used for the US unit system. Any text that follows an exclamation mark (!) denotes a comment and is ignored by GPS-X.

You may edit the US unit system, but **DO NOT change the SI unit system**. The unit in the US column **must be** compatible with the unit in the SI column. The compatible unit groups are listed in the second section of this file.

Adding New Unit Conversions

If you want to add new conversion factors to make them available in GPS-X, you can edit the second part of the file. You can either just add new units to existing groups or create your own groups.

This section, which constitutes the remainder of the file, starts at the line:

```
!CONVERSIONS:
```

The conversions part of the file is divided into sections of equivalent units (e.g., the flow units are grouped together).

Groups of similar units are separated by a blank line.

The first line in each group lists the default SI unit followed by the value of 1. **DO NOT edit this line** in the existing groups.

The remaining lines in each group contain units and their related conversion factors.

The default file contains a comprehensive, though not exhaustive, list of units. Some units are presented in multiple formats (i.e., million US gallons per day is given as `MGD(US)` and `Mgal/d(US)`). The user may wish to add extended ASCII characters to the file to improve the appearance of the units (i.e., for `m3/d` you may wish to provide an option for `m3/day`).

The conversion factor is what the new unit is multiplied by to convert it to the default SI unit.

For example, the unit conversion factor for `Mgal/d (US)` is `3785.41`, which means that `1 Mgal/d(US)` is equal to `3785.1 m3/d`.

Special Case

A special case format for unit conversions is used when the conversion between units is not a simple ratio. This is really only the case for temperature.

For example, to convert degrees Fahrenheit (F) to Celsius (C) requires the equation: **C = 0.55556 (F - 32)** which is represented by the conversion factor `0.55556/32` in the units file. Note that the slash (/) is not a divider but a delimiter that separates the multiplier from the constant.

CHAPTER 13

On-line Data Reading Tools

The GPS-X On-line Data Reading Tools are very powerful tools that are relatively easy to configure. In normal "off-line" mode, input data files are read only once at the start of a simulation. This requires the user to wait until all of the data that will be used in the simulation is collected, convert them manually to GPS-X `.dat` files, and then do the simulation runs. The ability to read data directly from on-line SQL databases is also described.

CONTINUOUS READING OF TEXT FILES

In on-line mode these `.dat` files are growing continuously, line by line, and GPS-X keeps monitoring and reading them at a specified sampling rate. If a new line appears in the data file, it will be brought in to GPS-X (the file input controller updated or the data appears as a new point on the graph) and the simulation advances to the new time. Thus, the simulation would be continuously updated with data arriving in real time from a plant SCADA system, and real time simulation of the plant would be achieved.

The user would see a data/model difference from the very moment it starts to emerge, indicating a problem. An additional tool called "sigtrack", found in the toolbox object, could be used to set up alarms if the data, simulation, or the error between them goes beyond certain limits.

An advanced on-line application (which would require the Dynamic Parameter Estimator, DPE) can do a real-time simulation and calibration/optimization simultaneously. In this way, the model calibration would be automated using the most up to date plant data.

To access the **On-Line** settings, select ***Layout* > *General Data* > *System* > *Input Parameters* > *Simulation Tool Settings***.

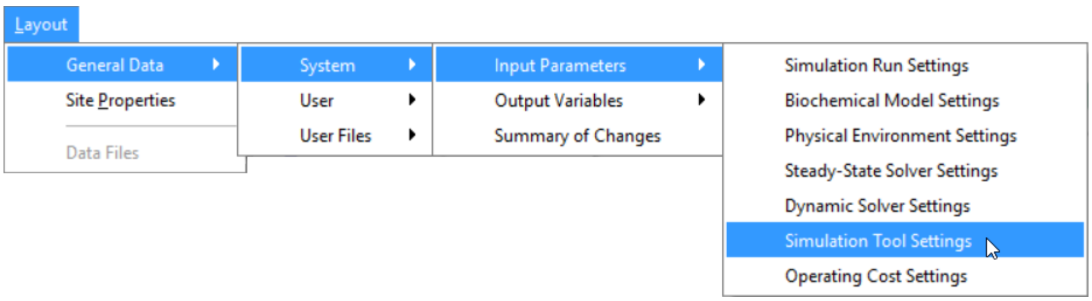


Figure 13-1 – Accessing On-line Operation Settings

Scroll down to the On-line Operation section. This is where you can turn on/off the on-line run mode (ie. continuous data reading and automatic simulation until the last point) and access additional settings under the **More** button.

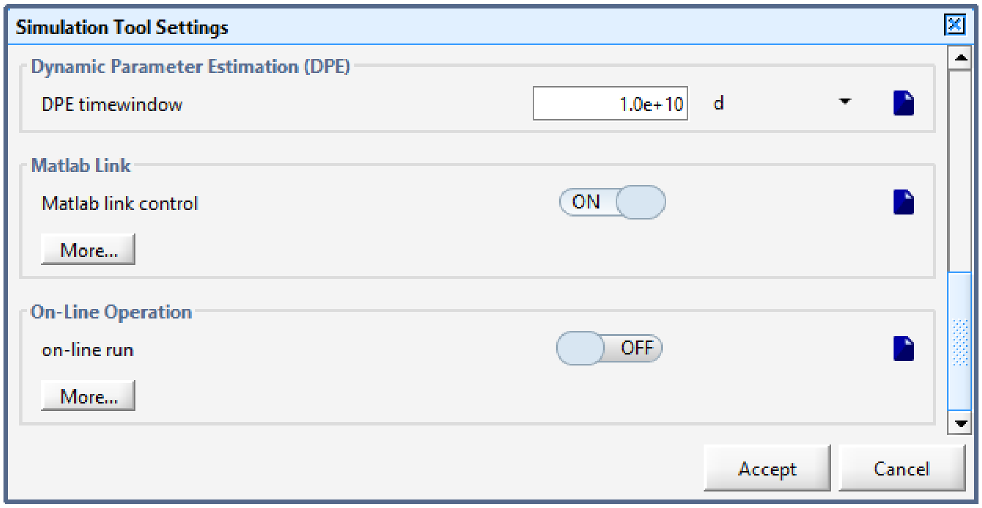


Figure 13-2 – Simulation Tool Dialog

The main on-line operation parameters include:

- **wait for all data to synchronize:** It is possible that not all data is coming in at the same rate. If ON, all of the variables on file input controllers must have data before the simulation will continue. This is meant to deal with small discrepancies in instrument logging frequencies.

- **waiting period:** The simulation will pause while it waits for data to accumulate in the input data file from the source (i.e. SCADA system). It will then continue only after data arrives that corresponds to at least the current simulation time plus the specified waiting period.

- **sampling rate from data base:** The file system is checked for new data at the specified rate. The default rate is 60 seconds.

Input data files are typically created by ‘bridge’ software that converts data from a format in a SCADA system to the format used by GPS-X. The format of online input data files is the same as the standard GPS-X input data files. For a complete explanation of the format see **Using File Input Controllers** in Chapter 6.

In on-line mode, it is possible to run multiple simulations concurrently and transfer data between simulations where the output from one layout is used as input file to another. (i.e. one layout optimizes settling, and then the next one uses optimized settling parameters). Typically, GPS-X reads data files with the naming convention <layoutname>_<>_<date>.dat and produces output files with <layoutname>_<>_<date>.out. To be able to use files generated by other layouts, GPS-X provides a way to specify non-default named files.

The user can enter the name of the layout that generates the .out file in the text field next to the **plant #1 name (for data file)** label. Click on **More…** to be able to add up to 10 file names. The layout would have to be rebuilt once the file names are entered (i.e. plant file names cannot be specified in a scenario).

One application of this functionality is to use a previously calibrated layout of a particular plant to generate real-time data that would otherwise be generated and read in from a real plant SCADA system. For example, if the previously calibrated layout name is *realplant.lyt*, then the **plant #1 (for data file)** field would be specified as *realplant*. The data generated by this realplant layout is then read in by a second layout running in a separate application of GPS-X according to the On-Line Operation settings in the second layout. In this way the user can simulate the on-line operation of GPS-X without having access to an actual SCADA system. The second layout will ignore any data files which are created after the simulation begins. Therefore, empty data files may need to be created prior to starting the online simulation to ensure that data is monitored and read in from the appropriate files.

USING AN SQL DATABASE WITH GPS-X

NOTE: The database feature requires the **"Advanced Module"** license. This feature also requires the appropriate JDBC driver for your database. GPS-X ships with drivers for the MySQL and Postgresql databases. JDBC drivers are readily available for other databases. Contact your database vendor for the driver for your system. To register any other driver with GPS-X, copy the appropriate JDBC jar file to ***/bin/gpsx/jar/*** directory and restart GPS-X.

After creating your layout, select ***File > Database Setup…***.

This will open a dialog where you can specify how to connect to your database, as shown in **Figure 13-3**. The form defaults to settings for the MySQL database system.

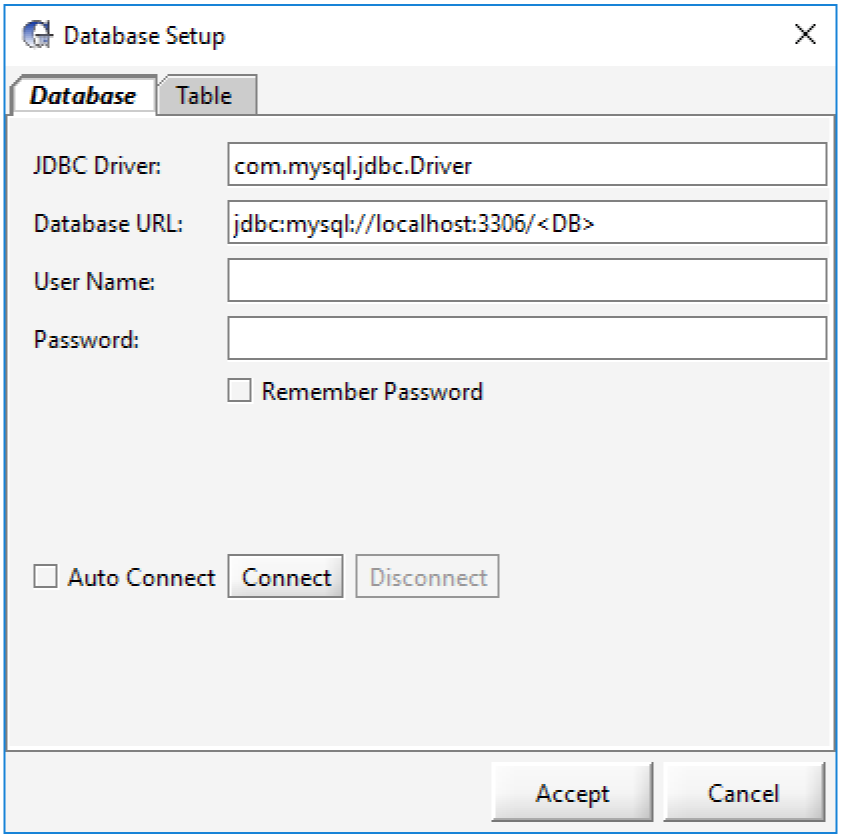


Figure 13-3 – Database Setup Dialog

JDBC driver: A unique driver path for each database system. Examples are:

- My SQL – com.mysql.jdbc.driver
- Postgresql - org.postgresql.driver

Database URL: Specifies which particular database to connect to and where it is located (local or remote) and port number. Examples are:

- MySQL - jdbc:mysql://localhost:3306/<DB>
- Postgresql - jdbc:postgresql://localhost:5432/<DB>

Replace localhost with an IP address or name if the database is not on the local system. Replace <DB> with the database name. If the database is on a remote system, make sure a firewall is not blocking the port for your database.

User Name and **Password** is required for your database system.

> NOTE: If you select Remember Password, the password is saved in clear text.

AutoConnect if you want the connection to the database to be opened when the layout is loaded.

Next, click on the **Table** tab. Here, you will specify the database table name where your data is stored along with mappings between database keys and GPS-X variable names. You can specify that the data is stored in a table that matches the current GPS-X scenario name or in a specific table, as shown in **Figure 13-4**.

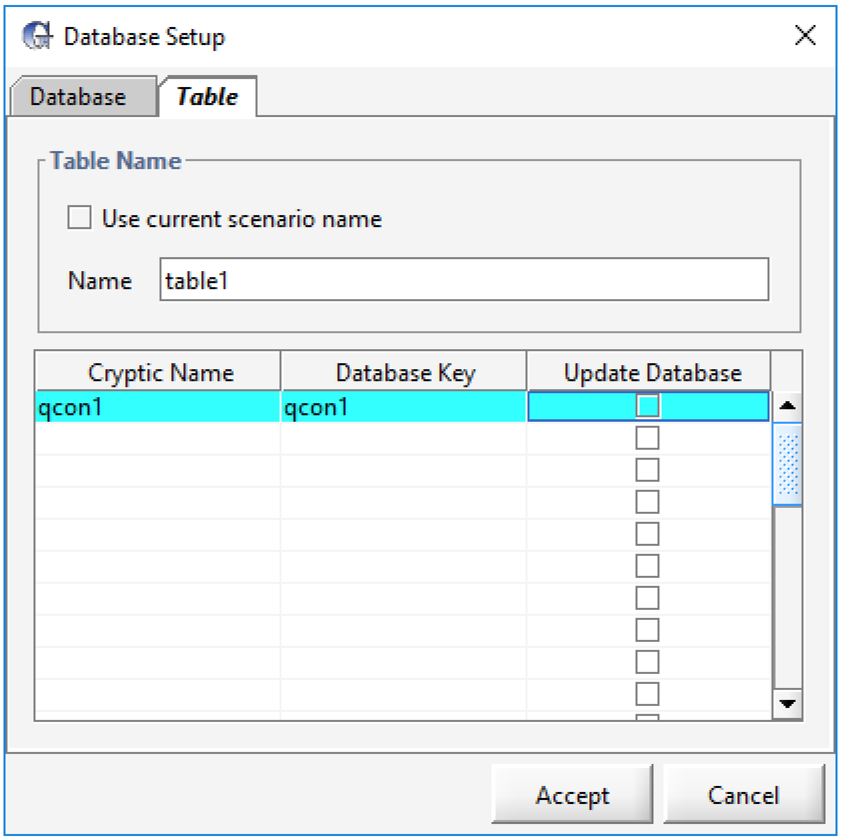


Figure 13-4 – Database Table Information Settings

The format of the data in the database table is time-stamped columns of data. The table's primary key is `'t'` (time in days), and each column represents one GPS-X variable (in SI units). The **Table Form** (see **Figure 13-5**) allows for the GPS-X cryptic name and database key to be mapped, if required. In the above example, they are identical.

t	qcon1
0.0	2300.0
0.1	4500.0
0.4	1400.0
0.7	2000.0
1.0	2500.0

Figure 13-5 – Table Form

After you have defined controls and graphs for your layout, right-click on the control and change the controller type to **"Database"** for the desired control item, as shown in **Figure 13-6**. The control will now work like the usual file input controller, except that the data will be read from your database.

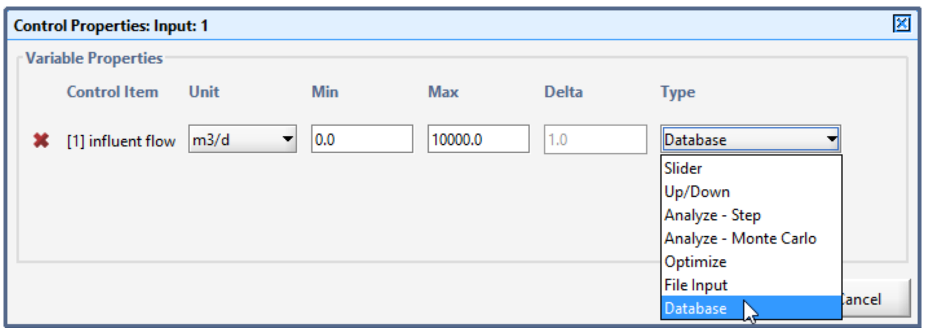


Figure 13-6 – Input Control Settings (selecting database)

If you require your database data to appear on a plot, access the **Output Properties** dialog and change the **Data Points** source to **Database** as shown in **Figure 13-7**.

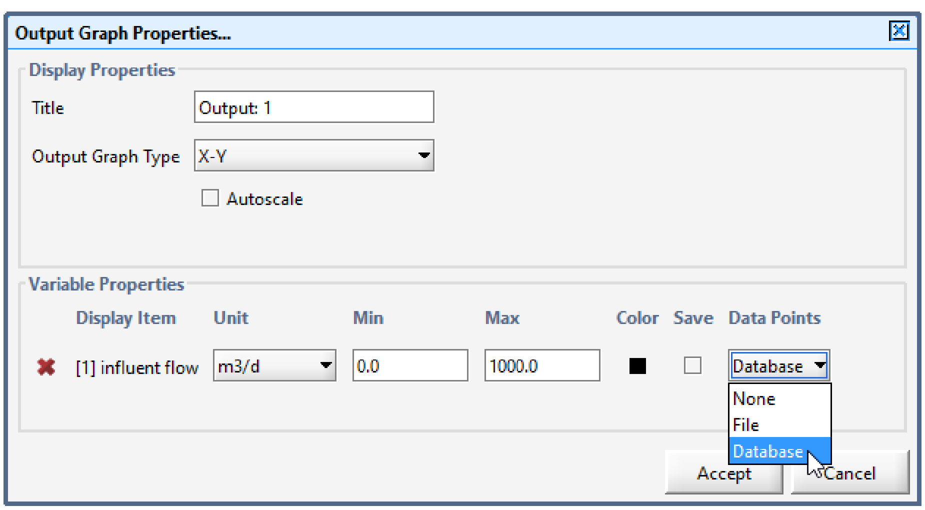


Figure 13-7 – Output Properties (displaying database data)

CHAPTER 14

GPS-X Python Integration

INTRODUCTION

Python integration directly into GPS-X allows you to build and run custom Python scripts directly within GPS-X. By allowing custom scripts to be run in GPS-X, you can use the tools available in Python to extend what is possible in GPS-X. Well the applications of GPS-X are diverse, it may not always be built generally enough to provide the exact analysis or data visualization options you would like for your project. By using Python, you can do things such as create custom plots, introduce new types of statistical analysis and automate repetitive tasks in ways that are not possible using only GPS-X.

This chapter covers the tools that have been built into GPS-X that allow you to control Python. The material includes an explanation of the Python dedicated menus in GPS-X as well as the custom Python commands that are recognized by GPS-X.

In general, this chapter assumes a more advanced level of understanding of both the GPS-X system itself as well as the Python programming. To use what is presented in this chapter, it is helpful if you have and understanding of the following:

- Python programing language
- Java programming language
- An understanding of how GPS-X preforms simulations

Python integration is an optional feature in GPS-X. If you do not have the Advanced Tools package, please contact Hydromantis for information about pricing.

THE PYTHON SCRIPT MANAGER

The Python Script Manager is used as the interface between GPS-X and Python. This tool is where you add, remove and run Python scripts in GPS-X.

The **Python Script Manager** is accessed through the ***Tools*** menu on the main menu bar. GPS-X must be in Simulation Mode to open the **Python Script Manager**.

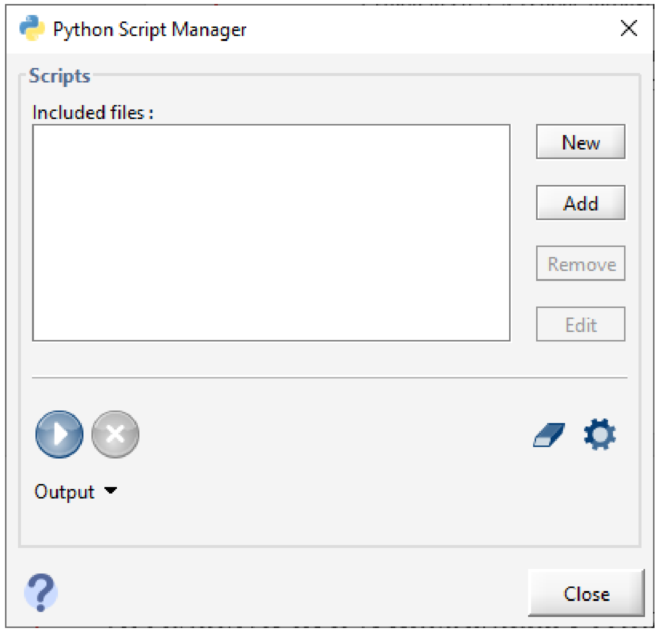


Figure 14-1 - The Python Script Manager

To create a new Python file in GPS-X, press the **New** button on the Python Script Manger. This will open a file manager window where you can choose where to save the script and what to name it. GPS-X will automatically apply the .py extension to the end of the file to signify that it is a Python Script file.

When a new file is created using the Python Script Manager, the path to the file will be added to the Included files section of the Python Script manager.

The Python Script Manager can also be used to regulate the Python script pathways available in the current layout by using the **Add** and **Remove** buttons.

EDITING PYTHON SCRIPTS

When a Python script is selected in the Python Script Manager, pressing the **Edit** button will open the Python script in the Notepad text editor by default. An unedited GPS-X Python script file produced by GPS-X can be seen in **Figure 14-2**.

In the Python file generated by GPS-X, four functions will be included that are recognized by GPS-X when a Python script is run by the Python Script Manager. These functions are:

start()

The start function is called exactly once in each simulation initiated by the Python Script Manager. The start function is executed at the start of the simulation before the first communication interval.

cint()

The cint function is called multiple times in a GPS-X simulation initiated by the Python Script Manager. The cint function will be executed at each communication interval in the simulation. A communication interval occurs at both the start and end of each simulation.

eor()

The eor function is called exactly once in each simulation initiated by the Python Script Manager. The eor function is executed at the end of the simulation after the final communication interval.

runSim()

The runSim function in Python initiates a GPS-X simulation. A new GPS-X simulation will be started each time the runSim function is called in the Python script.

For a complete list of all the Python functions recognized by GPS-X and their functionality, refer to **Appendix A** at the end of the Chapter.

UserGuide - Notepad

File Edit Format View Help

```
# Simple template for controlling GPS-X from a Python interpreter
# This script is launched from GPS-X
# GPS-X copyright 2019 Hydromantis

# start() function executed once at simulation start
#
def start():
 pass

# cint() function executed at every communication interval
#
def cint():
 pass

# eor() function executed once at end of simulation
# finished set True is required to terminate the runSim() function
#
def eor():
 global finished
 finished = True

# runSim() call starts simulation in GPS-X
try:
 runSim()
except Exception:
 pass
```

Figure 14-2 - Python Script File Generated by GPS-X Open in Notepad

RUNNING PYTHON SCRIPTS

Python scripts in GPS-X are run through the Python Script Manager. To run a Python script:

1. Ensure the Python script is selected in the Include files menu. The selected script is indicated by the file being highlighted blue.

2. Press the **Run Script** button to have the Python Script Editor execute the selected script.

3. At any time well the Python Script Manager is running a script, you can terminate the script by pressing the **Abort Script** button.

4. At the end of the simulation, any results printed in the Python script will be displayed in the Output menu of the Python Script Manager, which can be opened by clicking on the output arrow below the **Run Script** button. If an error is experienced while running the Python script, it will be displayed here.

   Additionally, output displays in GPS-X will retain the results of the simulation when the script has finished running, allowing you to navigate the simulation results in GPS-X directly.

Note: When rerunning a simulation the **Remove all Variables** button can be used to reset the variable values from memory.

PYTHON SETTINGS

In the **Python Script Manager** there is a button to access the various settings available for the Python instance used by GPS-X. This menu can also be accessed by going to *View > Preferences* on the main toolbar and selecting the Python tab.

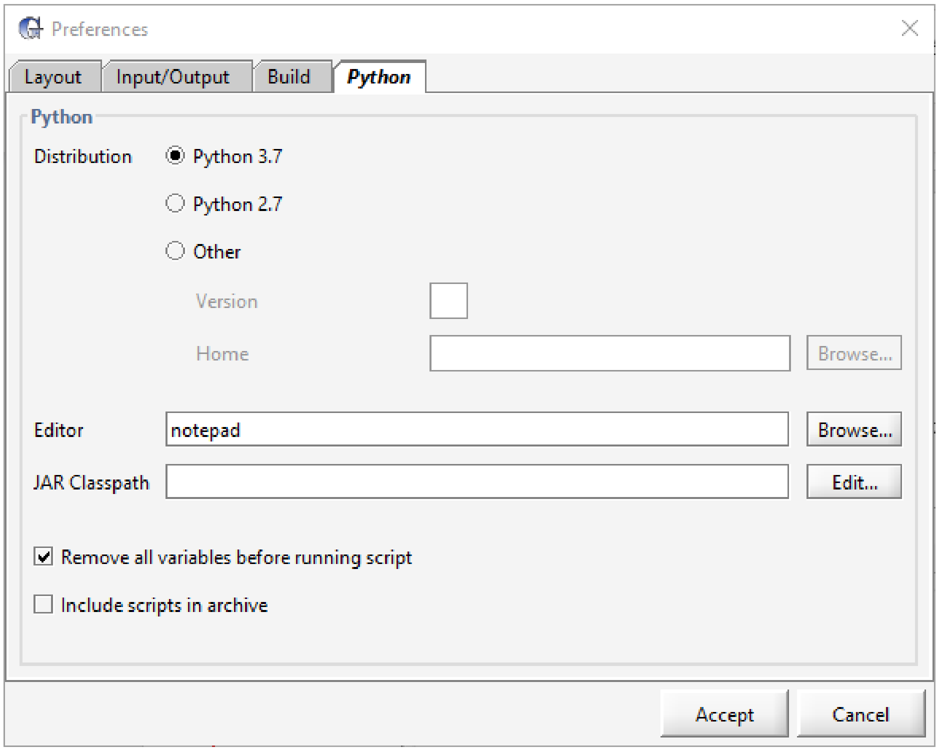


Figure 14-3 - Python Preferences Menu

Python Distribution

The user has the ability to specify which distribution of Python they would like to use in GPS-X. Python has two versions that are currently supported: Python 2 and Python 3. The GPS-X instillation includes both versions 2.7 and 3.7 of Python. By default Python 3.7 will be set as the active Python distribution used by the Python Script Manager. The user can easily toggle between these two distributions from this menu.

If you would like to use a Python distribution other than 2.7 and 3.7; selecting the other option will allow you to specify the distribution. You must provide GPS-X with a path to the instance of Python on your machine that you would like to use.

Editor

When using the Python Script Manager to edit Python Script files in GPS-X, the file be opened in the Notepad text editor by default. GPS-X has the ability to use any text editor you would like. To set a new default, press the **Browse** button next to the Editor entry field and provide GPS-X with a path to the Python Editor you would like to use.

JAR Classpath

There are instances where even Python is too limited to achieve what you would like. GPS-X accommodates the use of Java JAR files in Python. To include JARs, you must provide GPS-X a pathway to the JAR files. To do this, press the **Browse** button next to the JAR Classpath entry field. You can provide GPS-X with either a path to a single JAR file or a folder of multiple JARs.

APPENDIX A: GPS-X RECOGNIZED PYTHON FUNCTIONS

Table 14-1 - GPS-X recognised Python Functions

Function	Parameters	Description
gpsx.getVersion()	–	Returns the current version of GPS-X as a string
gpsx.getLibraryName()	–	Returns the current library name as a string
gpsx.iconize()	–	Minimize the GPS-X interface
gpsx.getLytName()	–	Returns the current layout name as a string
gpsx.getSigDigits(ndigits)	ndigits: *int*	Sets the number of Significant Digits GPS-X reports
gpsx.createReport(name)	Name: *string (optional)*	Generates an .xls report. If no name is specified, the file will be saved with the name (layoutnme+scenarioname.xls)
gpsx.exit(error)	error: *int (optional)*	Forces the GPS-X interface to exit. If no error code has been specified, it will exit with error code 0.
gpsx.setScenario(scenario)	scenario: *string*	Set GPS-X to use the specified scenario. The scenario can not be changed once the simulation has started

gpsx.sendCommand(command)	command: *string*	Execute a command in the GPS-X command window
gpsx.setTstop(tstop)	tstop: *int*, *float*	Set the simulation duration to the value of *tstop* in days
gpsx.setCint(cint)	cint: *int*, *float*	Set the simulation communication interval to the value of *cint* in days
gpsx.setDelay(delay)	delay: *int*, *float*	Set the simulation delay to the value of *delay* in seconds
gpsx.setSteady(steady)	steady: *boolean*	Enable/Disable steady state in the simulation. Steady State can not be changed once the simulation has started
gpsx.writeToCommandWindow(text)	text: *string*	Write the text to the command window
gpsx.resetAllValues()	–	Resets all variables back to their values from when first entering simulation mode
gpsx.resetSim()	–	Resets simulation mode
gpsx.startSim()	–	Starts the simulation. This is an alternative to the runSim() function, but runSim() is preferred
gpsx.interruptSim()	–	Interrupts the running simulation

gpsx.setAnalyzeMode(type)	type: *string* One of: ('steady state', 'phase dynamic', 'time dynamic', 'montecarlo')	Set the simulation to analyze mode. You must specify one which Analyze method you would like to use.
gpsx.setOptimizeMode(type, objfun)	type: *string* One of: ('Time Series', 'DPE') objfun: *string* One of: ('Absolute Difference', 'Relative Difference', 'Sum of Squares', 'Relative Sum of Squares', Maximum Likelyhood')	Sets the simulation to optimize mode. You must specify the type of optimization you would like to preform and the objective function you would like to use.
gpsx.setValue(cryptic, value, units)	cryptic: *string* value: *int, double, Boolean, string* units: *string (optional)*	Set the value of a layout variable. The unit value specifies the units of the value you are entering and must use the same format as GPS-X ie 'MGD(US)'. If units are not entered, it is assumed that the data is in the default units of US/SI units set in the GPS-X interface.
gpsx.setValueAtIndex(cryptic, index, value, units)	cryptic: *string* index: *int* value: *int, double, Boolean, string* units: *string (optional)*	Set the value of an element of an array at the specified index. The unit value specifies the units of the value you are entering and must use the same format as GPS-X ie 'MGD(US)'. If units are not entered, it is assumed that the data is in the default units of US/SI units set in the GPS-X interface

gpsx.getValue(cryptic, units)	cryptic: *string* units: *string (optional)*	Gets the value of a layout variable. The unit value specifies the units the value will be returned in ie ‘MGD(US)’. If units are not entered, it returns the value in the default units of US/SI units set in the GPS-X interface.
gpsx.getValueAtIndex(cryptic, index, units)	cryptic: *string* index: *int* units: *string (optional)*	Gets the value of an element of an array at the specified index. The unit value specifies the units the value will be returned in ie ‘MGD(US)’. If units are not entered, it returns the value in the default units of US/SI units set in the GPS-X interface.
gpsx.getArrayValues(cryptic, n, units)	cryptic: *string* n*: int (optional)* units: *string (optional)*	Gets all the values of an array variable and returns them as a list. If a value of n is specified, the function will return the first n elements of the array. The unit value specifies the units the value will be returned in ie ‘MGD(US)’. If units are not entered, it returns the value in the default units of US/SI units set in the GPS-X interface.
gpsx.createAnalyzeCode()	–	Creates code to perform an analyze mode run by scanning for controllers set to the Analyze control type. Analyze results will be copied to the Windows clipboard