Series 4000 GAS MIXING SYSTEM
Series 4040 GAS DILUTION SYSTEM

User Manual

Software Revision 1.2.x
Revised 2017-10-04

Environics, Inc.
69 Industrial Park Road East
Tolland, CT 06084-2805 U.S.A.
Phone: (860) 872-1111 Fax: (860) 870-9333
E-mail: INFO@ENVIRONICS.COM **Web:** HTTP://WWW.ENVIRONICS.COM

COPYRIGHT
© 1996-2009 Environics, Inc. All Rights Reserved. This manual and the software contained within the product(s) described are copyrighted with all rights reserved.

TRADEMARKS
Environics is a registered trademark of Environics, Inc. All other brand names, company names and product names mentioned are the property of their respective owners.

WARRANTY
Environics, Inc. warrants this product to be free from defects in material and workmanship for a period of one year from the date of shipment. Environics warrants the following expendable items for 30 days from the date of shipment: fuses, lamps, batteries. During the warranty period, Environics will, at our option, either repair or replace any product that proves to be defective.

To exercise this warranty, contact Environics at the address below for assistance and instructions for returning the products. Repaired or replaced products are warranted for the balance of the original warranty period or at least 30 days.

LIMITATION OF WARRANTY
This warranty does not apply to defects resulting from product modification made without Environics' express written consent, or misuse of any product or part. This warranty also does not apply to software, damage from battery leakage or problems arising from normal wear or failure to follow instructions.

This warranty is in lieu of all other warranties, expressed or implied, including any implied warranty of merchantability or fitness for a particular use. The remedies provided herein are the buyer's sole and exclusive remedies.

Neither Environics nor any of its employees shall be liable for any direct, indirect, special, incidental or consequential damages arising out of the use of its instruments and software even if Environics has been advised in advance of the possibility of such damages. Such excluded damages shall include, but are not limited to: costs of removal and installation, losses sustained as the result of injury to any person or damage to property.

WARNING
READ THIS MANUAL CAREFULLY BEFORE USING THIS INSTRUMENT.
FAILURE TO DO SO MAY VOID THE WARRANTY,
DAMAGE THE INSTRUMENT OR CAUSE SERIOUS INJURY.

Environics, Inc.
69 Industrial Park Road East
Tolland, CT 06084-2805 U.S.A.
Phone: (860) 872-1111 Fax: (860) 870-9333
E-mail: INFO@ENVIRONICS.COM **Web:** HTTP://WWW.ENVIRONICS.COM

S4000/4040 User manual changes

Manual Revision #	Software Revision #	Manual Release Date	Enhancements
1			Initial release
2	1.2.x	2009-03-18	Added Instrument Data installation from CD
			Added information about serial/USB adapters
			Added quick start info from TB #131
3		2017-10-04	Updated Windows operating system requirements

SERIES 4000 GAS MIXING SYSTEM
SERIES 4040 GAS DILUTION SYSTEM

ENVIRONICS, INC
69 INDUSTRIAL PARK ROAD EAST
TOLLAND, CT 06084

EQUIPMENT SUMMARY – TYPE: (X) Measurement ( )Control ( ) Laboratory

Marked electrical rating:
115 VAC, 50/60 Hz, 2A
230 VAC, 50/60 Hz, 1A

Connection to Supply: Standard IEC Inlet Coupler/Cord Connected

Environmental rating:	**Standard**
Operating Temperature:	**0-50°C**
Performance temperature range	**15-35°C**
Humidity:	**<95% RH @ 15-30 deg C**
Altitude:	**up to 2000 Meters**

Installation Category II
Pollution Degree: 2

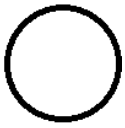
Equipment Class I

Equipment function/feature/description:
S4000/4040 was designed for precision mixing/dilution of gases.

Mode of operation: Concentration Mode, Flow Mode, Program Mode, Calibrate Mode.

Degree of mobility: ( ) Permanently Connected (X) Other Cord Connected.

Overall size: 17”x7”x15” Weight: 27 lbs.

SYMBOLS USED	PUBLICATION	MEANING
	ISO 3864, No. B.3.1	CAUTION (Refer to accompanying documents). Refer to specific WARNINGs and CAUTIONs in Operating Manual.
	IEC 417, No. 5019	PROTECTIVE CONDUCTOR TERMINAL
	IEC 417, No. 5007	ON (SUPPLY)
	IEC 417, No. 5008	OFF (SUPPLY)

POWER CONNECTION

The main power disconnect for the unit shall be the power cord that is plugged into the rear of the unit. The instrument is totally enclosed at all times with a top and bottom cover for safety.

Position the equipment so that it is not difficult to operate the disconnecting device.

Table of Contents

1. INTRODUCTION 1

2. SETUP AND CONFIGURATION 3

2.1 Connecting to a PC 3
2.2 System Software Installation 3
2.3 Instrument Data Installation 4
2.4 Operating Manual 4
2.5 Quick Start Guide 5

3. RUNNING THE SOFTWARE 6

3.1 Main Menu Overview 6
3.2 Summary Information 7

4. MANAGING DESKTOP CONFIGURATIONS 8

4.1 Creating a new desktop 8
4.2 Opening an existing desktop 8
4.3 Assigning a different instrument to an existing desktop 9
4.4 Deleting an existing desktop 9
4.5 Closing a desktop 9

5. BASIC CONFIGURATION 10

5.1 Gas Library Configuration 10
5.2 Cylinder Configuration 12
5.3 Port Configuration 15
5.4 Instrument Configuration 16
5.5 Serial Port Configuration 17
5.6 Cylinder Auditing 17
5.7 Log File Configuration 18

6. RUNNING THE INSTRUMENT 19

6.1 Concentration Mode 19
6.2 Flow Mode 23
6.3 Divider Mode 26
6.4 Program Mode 30
6.5 Schedule Mode 33

7. GENERATING REPORTS 37

8. APPENDIX 39

8.1 Example 1 - Blending Gases 39
8.2 Example 2 - Diluting Gases (4040 only) 41
8.3 Example 3 - Dividing Gases 43
8.4 MFC Calibration Procedure 45
8.5 Working with multiple calibration tables 46

1. Introduction

This manual describes the setup and operation of the Environics Series 4000 Computerized Gas Mixing System and Series 4040 Computerized Gas Dilution System. Detailed operating instructions for the Environics Series 4000 software in also included.

The system consists of two primary components: The Environics Series 4000 / 4040 instrument and the Environics Series 4000 software running on the user's PC. The Series 4000 software acts as the user interface to the instrument.

The PC communicates with the instrument via an RS232 serial interface. The instrument contains minimal control capabilities as a stand-alone unit and requires the PC for all primary control. The stand-alone capabilities are limited to the ability to maintain the current port and flow controller flow settings at the time the interface cable is disconnected. In addition, the instrument front panel contains an on/off switch and a flashing green LED status indicator.

The Environics Series 4000 software is a 16-bit Microsoft Windows application, and will run on Windows 95/98/2000 and Windows XP. It can also be run on Windows 7/8/10 in Windows XP mode.

2. Setup and Configuration

2.1 Connecting to a PC

The PC communicates with the S4000 through the supplied RS232 null modem cable. Connect one end of the cable to the RS232 connector on the rear of the S4000, and connect the other end of the cable to any available serial port on the PC.

If the PC is not equipped with a serial port (COM port), a USB/Serial adapter can be used. Plug the USB adapter into any available USB port. Connect one end of the nulll modem cable to the RS232 connector on the rear of the S4000, and connect the other end of the cable to the serial port on the USB adapter.

When using a USB adapter, it must be configured to use any COM port 1 through 4. To change the USB adapter COM port setting:

- Right click My Computer, and click Properties
- Click the HARDWARE tab, and then click DEVICE MANAGER
- Click the + icon to expand the PORTS list
- Locate the listing for your USB-serial adapter, and double-click it.
- Click the PORT SETTINGS tab, then click ADVANCED
- Change the COM PORT NUMBER value to between COM1 and COM4, then click OK

Refer to the USB/Serial adapter installation instructions for more informationon how to configure the adapter COM port.

The S4000 software uses PC COM port 1 by default, but supports any COM port 1 through 4. The COM port used may be changed from the CONFIGURE menu of the S4000 software.

The enclosed Series 4000 CD contains the S4000 software, instrument data specific to this system and the S4000 operating manual in PDF format.

2.2 System Software Installation

The software installation process installs the main application program and required support files. The system software needs to be installed only once, regardless of the number of instruments purchased.

NOTE: *For some PC operating systems, it may be necessary to grant the user full Administrator rights to install and operate the Series 4000 software. In particular, the user needs full read/write access to the following folders: C:\ENV4UI C:\WINDOWS C:\WINDOWS\SYSTEM*
Consult your system administrator for more information about setting user access rights.

1. Close all running Windows applications.
2. Insert Environics Series 4000 Software CD-ROM in to the CD Drive
3. If CD Autorun is enabled, the installation program will start automatically. Otherwise, click START, RUN, and enter "D:\SOFTWARE\SETUP". (replace D: with the drive letter for the CD Drive).

   a) For Windows 3.1, select FILE, RUN instead of START, RUN

IMPORTANT: *If you are installing a new version of the software over a previous version and want to preserve your existing data, you must click the INSTALL OPTIONS button and uncheck the INITIALIZE DATABASE box. Failure to uncheck this box will overwrite the existing configuration database with a blank database, which will erase all user settings including Gases, Cylinders and Flow/Concentration/Program mode settings.*

4. Follow the on-screen instructions. First time users should not change the default INSTALL OPTIONS
5. To run the software, click Start, Environics, Environics Series 4000, or double-click the Environics Series 4000 icon.

2.3 Instrument Data Installation

Instrument Data for each instrument is supplied on the CD. The instrument data contains detailed calibration and configuration data that is unique to each instrument. The instrument data for each physical instrument must be installed before it can be operated.

1. Be sure the Series 4000 CD is in the CD drive.
2. Start the Environics Series 4000 software
3. From the Main Menu, select *Configure* - *Instrument* - *Install*.
4. When the Install Instrument box appears, select the appropriate CD drive icon (typically D: or E:)
5. In the "Path to Installation Disk" window, double-click on the DATA folder.
6. Locate the folder with the Serial # of your instrument, and double-click it.

   IMPORTANT: Be sure to DOUBLE-CLICK the serial # folder (this will make the folder appear "open"). If the folder is not "open", you will get an error when installing the instrument data.

7. Click *OK* to install the instrument data. If the instrument data is installed successfully, no messages will be displayed, and you will be returned to the Environics Series 4000 Main Menu

   NOTE: If you have ordered your systems Mass Flow Controllers to be calibrated in more than one gas, there will be additional subfolders with gas names under the serial # folder containing additional Instrument Data (1 subfolder per each gas calibration). Repeat the process above for each gas subfolder. To switch calibration tables, refer to Section 8.5 - Working with multiple calibration tables

2.4 Operating Manual

The Series 4000 operating manual is located on the CD in Adobe PDF format (Adobe Acrobat Reader is required). The manual is located on the CD in the \MANUAL folder. Additional information about updates to the software and other special notices can be found on the CD in the READ_ME.TXT file.

IMPORTANT: If the unit was accompanied by a system operator addendum, please read carefully before running gases through the unit. There will be specific details to the setup and operation of the unit based on its unique configuration.

2.5 Quick Start Guide

This quick start guide gives an abbreviated set of instructions for getting the S4000 system up and running quickly. More details about complete operation of the system are provided starting in Section 3 - Running the Software

1) Make sure unit is connected to PC via supplied null modem RS-232 cable (Refer to Section 2.1)
2) Power up unit from front power switch. Make sure green status light is blinking.
3) Connect gas cylinders to ports. The balance gas or dilution gas will be connected to port 1.
4) Set input supply pressures to approximately 20-25 psig
5) Connect output line from OUTPUT bulkhead on rear panel to measurement device
6) Refer to software instructions to setup and run the system (Refer to Sections 4-5 of the manual)
    a. Create a new DESKTOP (Ref. Section 4.1 of the manual)
    b. Create the cylinders that will be connected to the rear panel (Ref. Section 5.2 of the manual)
    c. Assign the cylinders to specific ports as they will be connected physically to the rear panel (Ref. Section 5.3 of the manual)
7) Example for running a dilution in CONC mode:
    a. From an open desktop, click RUN, then CONCENTRATION
    b. Create a new name in the CONC mode by clicking ADD then naming the particular setup
    c. Use the pull down arrows in the table to select CYLINDERS that have been assigned to the ports
        - Port 1 will be used as the balance/dilution gas and should go on the top line in the table
        - Port 2 will typically be the cylinder you wish to dilute
    d. Make sure the red check mark in the BALANCE column is on the line corresponding to the dilution gas on port 1.
    e. On the target cylinder line, change unit value to match desired target concentration (ppm or %)
    f. Enter a target value of the cylinder connected that is desired at the output.
    g. Enter a total flow rate in the TOTAL FLOW field that corresponds to the maximum size of MFC1.
    h. Click RUN to start the unit flowing. The feedback flow column should show values close to the target values.

3. Running the Software

The Environics Series 4000 software is installed in the Environics program group. The Series 4000 software is started by any standard Windows method, normally by double clicking the icon with the mouse. Double clicking the program icon shows the startup screen, which indicates that the program has started and is initializing.

Once started, the Series 4000 main window is displayed, as shown in Figure 1. The main window consists of a menu across the top, the Series 4000 instrument logo, and summary information across the bottom.

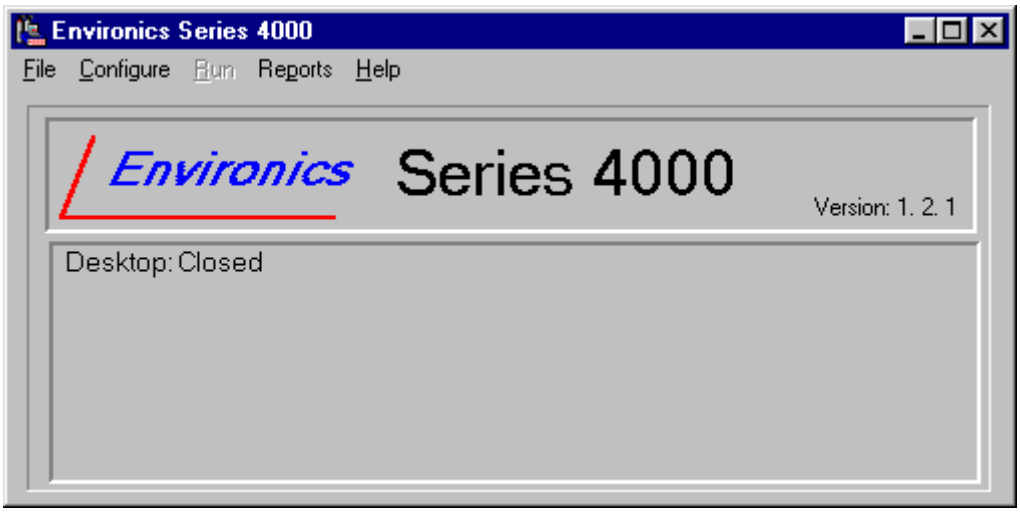


Figure 1 - Series 4000 Main Window

3.1 Main Menu Overview

The major system functions are grouped together in the Main Menu. The function groups are:

File: This menu item gives access to functions used to manage instrument desktops. A *desktop* is a stored set of instrument settings required for a given task. Desktops allow the configuration settings for a set of related tasks to be grouped together. The functions in the File menu allow desktops to be managed as follows:

New:	Opens a new Desktop.
Open:	Opens an existing Desktop.
Delete:	Deletes an existing Desktop.
Close:	Saves and closes the currently opened Desktop.
1/2/3/4:	Shows the 4 most recently opened Desktops, allowing them to be opened quickly.
Exit:	Exits the Environics Software.

These functions are discussed further in *Section 4 – "Managing Desktop Configurations".*

Configure: This menu item gives access to all functions used to manage basic instrument and library configuration. This includes definitions of gases, cylinders, and port connections. The configuration functions are:

Port:	Opens the Port Configuration form.
Cylinder:	Opens the Cylinder Library form.
Gas:	Opens the Gas Library form.
Instrument:	Contains a list of functions for configuring the instrument as follows:
- *Install*:	Allows installation of new instruments in the database.
- *Controller Options*:	Opens the controller options form.
- *Calibrate*:	Opens the instrument calibration form.
Select Serial Port:	Selects which PC COM Port is to be used by the software.
Cylinder Auditing:	Sets up warnings for cylinders that are beyond their expiration date.
Log File:	Sets up a log file for generating audit trail logs.

These functions are discussed further in *Section 5 –"Basic Configuration".*

Run: This menu item gives access to the following instrument run modes:

Concentration:	Opens the Concentration mode setup and run form.
Flow:	Opens the Flow mode setup and run form.
Divider:	Opens the Divider mode setup and run form.
Program:	Opens the Program mode setup and run form.
Schedule:	Opens the Schedule mode setup and run form.

Note: The RUN menu is not accessible unless a Desktop is open.

These functions are discussed further in *Section 6 – "Running the Instrument".*

Reports: This menu item gives access to the various instrument reports. Report can be viewed on the screen, printed, or saved to a file.

These functions are discussed further in *Section 7- "Generating Reports".*

Help: This menu item gives access to the instrument help functions. These functions provide on screen documentation on how to use the instrument and customer support information.

3.2 Summary Information

The summary information given at the bottom of the main window includes:

Version:	Displays the version number of the software.
Desktop:	Displays the name of the currently opened desktop or "CLOSED" if no desktop is open.

When a desktop is open, the main window also displays the following information:

Instrument:	The serial number of the instrument assigned to the desktop.
Port assignment:	A list of cylinders currently assigned to the instrument's gas ports.

4. Managing Desktop Configurations

A *desktop* is a stored instrument setup required for a given task. Desktops allow the instrument configurations for related tasks to be grouped together and saved. Using a different desktop for distinct tasks allows instrument settings to be quickly switched without having to manually re-enter settings related to that task. The following setup information is stored in the desktop file.

- All Cylinder/Port assignments.
- All Concentration, Flow, and Divider mode setups.
- All Program and Schedule mode setups.

NOTE: A desktop must be open in order to access the instrument RUN modes, or any operation described above which is specific to a desktop.

Some information is NOT stored along with a given desktop configuration. This information is common to all desktops and modifications made in one desktop will be reflected in all other desktops. This includes the following:

- Gas Library
- Cylinder Library
- Instrument Configuration and Calibration Settings

The functions for managing desktops are listed as entries in the File menu on the main window. These functions allow Desktops to be managed as follows:

New:	Creates a new desktop.
Open:	Opens an existing desktop
Delete:	Deletes an existing desktop.
Close:	Closes the currently opened desktop.
Exit:	Exits the software.

In addition, a list of the four most recently used desktop is displayed at the bottom of the File menu. Selecting a name from this list opens the desktop.

To begin using the Environics Series 4000 Software, you must create a new desktop file or open an existing one. All set-up information and configurations will be saved in this file.

4.1 Creating a new desktop

1. Select *File* and then *New* from the main menu window.
2. In the DESKTOP box, enter a name for the Desktop. The name can be up to twenty characters long.
3. In the INSTRUMENT box, click the drop down arrow to display the list of available instruments. Select the serial number of the instrument you wish to use.
4. Click *OK* to save the new file.

Note: If you obtain a warning that there are no instruments registered, you must first perform the steps in Section 2.3 - Instrument Data Installation.

4.2 Opening an existing desktop

1. Select *File* and *Open* from the main menu.
2. Click on the name of the file to open.
3. Click *Open* to open the selected file.

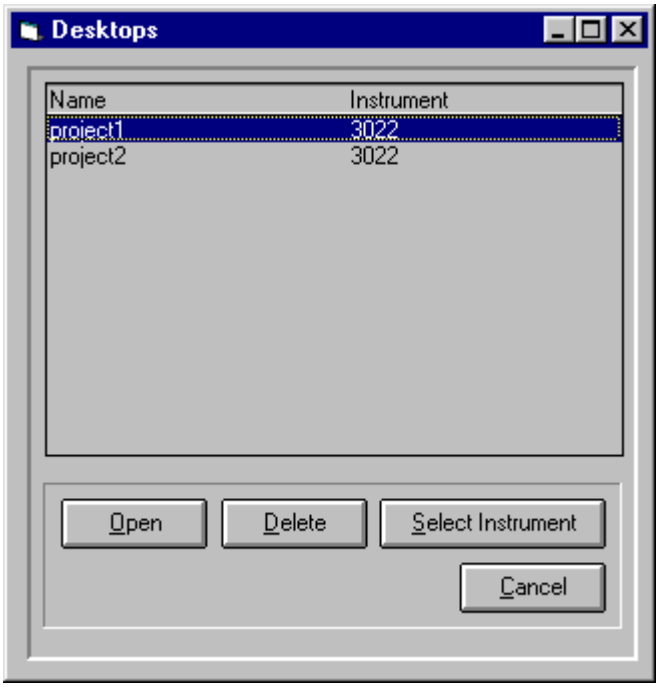


Figure 2 - Desktop selection window

4.3 Assigning a different instrument to an existing desktop

1. Select *File* and *Open* from the main menu.
2. Click on the name of the file to edit.
3. Click on *Select Instrument*. Use the pull down menu to select a new instrument.
4. Click the *OK* button to return to the Open menu.
5. Click *OK* to open the file or *CANCEL* to return to the main menu without opening.

Note: When opening a desktop, the software will try to communicate with the assigned instrument. If it is unable to communicate, a warning message will be displayed. The software also checks the serial number of the connected instrument to be sure it matches the instrument assigned to the desktop. If the serial numbers do not match, a warning message is displayed.

4.4 Deleting an existing desktop

1. Select *File* and *Delete* from the main menu.

Click on the name of the file to delete.
Click *Delete* to remove the file from memory.
Click the *Yes* button to confirm the deletion and return to the main menu.

4.5 Closing a desktop

Select *File* and *Close* from the main menu. The currently open desktop file will be closed.

Note: The current desktop file is automatically closed upon exiting the software. The desktop file must be reopened the next time you run the software.

5. Basic Configuration

The basic configuration functions are used to enter configuration information as follows:

- Gas, Cylinder, and Port definitions that are referenced by the instrument run modes.
- Instrument Calibration and Controller options.
- Serial port setting.
- Cylinder Auditing.
- Log file setup.

This information is common to all desktops. Modifications made will be reflected in all desktops. Each of these configurations is discussed in detail in the sections that follow.

5.1 Gas Library Configuration

The Gas Library defines the list of gases and corresponding physical characteristics that are used by the instrument. Gases are referenced primarily when defining cylinders and setting up run modes. The software comes with a library of over 100 commonly used gases, and includes the ability to create your own user-defined gases.

From the Main menu, select Configure – Gas to open the Gas Library (shown in Figure 3).

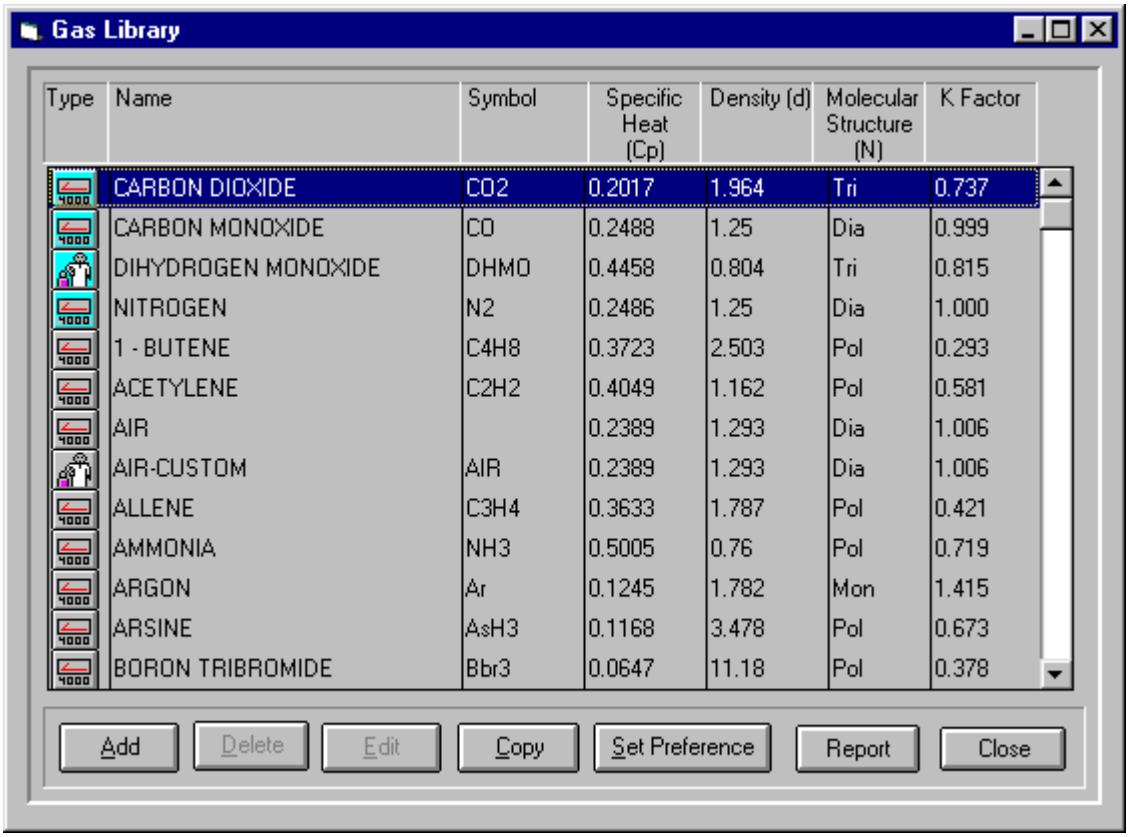


Figure 3 - Gas Library

The Gas Library consists of a list that shows the current gases in the library. The entry for each gas in the list shows:

Type:	The type of gas (System or User-defined). System gases are identified by the "4000" symbol.
Name:	The name of the gas.
Symbol:	The molecular symbol of the gas.
Specific Heat (C_p):	The specific heat of the gas in (cal / g ºC @ 25 ºC)
Density (d):	The density of the gas (g / l @ 0 ºC)
Molecular Structure (N):	The molecular structure of the gas: (Monatomic, Diatomic, Triatomic, or Polyatomic)

User-defined gases are edited by pressing one of the four buttons (Add, Delete, Edit or Copy) as follows:

Add:	Adds a user-defined gas to the gas library
Delete:	Deletes a user-defined gas from the library.
Edit:	Modify an existing user-defined gas.
Copy:	Copy the data from the currently selected gas to a new gas.
Set Preference:	Toggle the "preferred" status of the gas. If the gas is preferred, it appears at the top of the gas library for easy selection.

Note: Only user-defined gases can be edited/deleted.

When Adding/Editing a gas, the following information is required in the Gas Edit form:

Gas Name:	Enter the name of the gas
Gas Symbol:	Enter the symbol for the gas
Specific Heat:	Enter the specific heat value of the gas
Density:	Enter the density value of the gas
Molecular Structure:	Select the molecular structure of the gas from the pull down list.
Preferred:	Select this box to display this gas at the top of the gas list.

Adding a user defined gas

Click the *Add* button. A blank Gas Edit form will be opened. Enter the Gas Name, Gas Symbol, Specific Heat, Density, and Molecular structure (using pull down box).
Click the *OK* button to add the new gas to the library

Modifying a user defined gas

Select the gas to be edited by clicking on it.
Click on *Edit* to modify the currently selected user defined gas.
After modifying the data, click *OK* to save the modifications.

Deleting a user defined gas

1. Select the gas to be deleted by clicking on it.
2. Click the *Delete* button to remove the currently selected gas from the gas library. The gas is deleted immediately without a confirmation prompt.

Making a copy of a gas

Select the gas to be copied by clicking on it.
Click the *Copy* button to copy the data for the currently selected gas to a new gas. This opens the Copy Gas window. The software adds an asterisk "*" to the gas name and symbol, to differentiate it from the original gas. This name/symbol can be changed to something else if you prefer, so long as they are different from the original. You can also change the values that define the gas at this time if you wish.
Press the *OK* button to create the new gas.

Changing the order in which gases are displayed

Gases are normally displayed in alphabetical order. If certain gases are used frequently, you can give them "preferred" status. All "preferred" gases are displayed at the top of the gas library for easy selection.

To change the gases preferred status:
1. Select the gas by clicking on it.
2. Click the *Set Preference* button to toggle its preferred status on or off.

Generating Reports

To generate a report for one or more gases in the library, press the *Report* button. This report can be displayed on the screen, printed, or saved in a file. See Section 7 - Generating Reports for further details.

Saving Changes

When all desired Gas Library changes are made, press the *Close* button to return to the main menu window.

5.2 Cylinder Configuration

The Cylinder Library maintains a list of all cylinders that will be used in mixing or diluting gases. The Cylinder Library must be set up before the input Port Assignments can be defined. An individual cylinder consists of one or more component gases. Any gas in the Gas Library is available to be added to a cylinder. Once created, cylinders are used for defining input port connections and setting up the instrument run modes.

To open the Cylinder Library, select *Configure - Cylinder* from the main menu. This will open the Cylinder Library form (Figure 4) for the first cylinder in the library, or a blank form if the library is empty. Existing cylinders definitions can then be modified or new ones can be added.

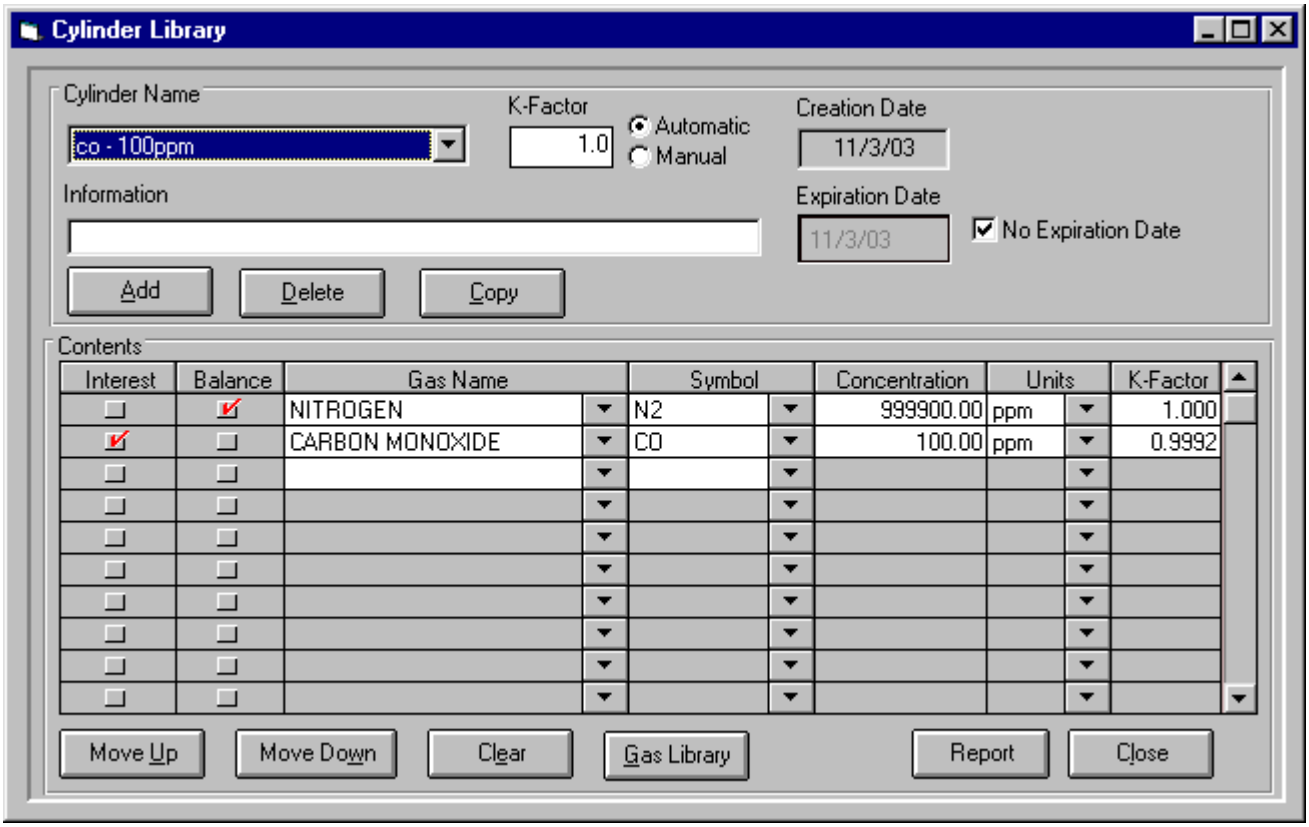


Figure 4 - Cylinder Library form

The Cylinder Library form consists of the following information:

Cylinder Name: This is the name of the cylinder. You can assign any name you want, but it is best to be descriptive (ex. "100ppm CO2 bal N2"). The currently selected cylinder is displayed here. To select a different cylinder, click on the pull down box and select the desired cylinder from the list by clicking on it.

Information: This is a user-defined field for storing any desired information about a cylinder (serial number, description, etc). The software does not use this information.

K-Factor: Gas correction factor of the cylinder. This is calculated automatically based on the component gases and their concentrations, unless set to MANUAL. When set to Manual, the user can enter the desired K-factor.

Creation Date: This displays the date that the cylinder was created. This value is generated automatically by the software and is not editable.

Expiration Date: This displays the expiration date of the cylinder. When used with the Cylinder Auditing feature (see Section 5.6 below), the software will warn you about cylinders that are approaching or are beyond their expiration date. By default, newly created cylinders have no expiration date. To insert an expiration date, uncheck the "No Expiration Date" box and enter the desired expiration date.

Understanding Cylinder K-Factors

The K-factor (or gas correction factor) describes the unique thermal characteristics of the gases in the cylinder. Since different gases have different thermal properties, the K-factor accounts for the differences between the gases being used in the system and the gas used during calibration of the Mass Flow Controllers (typically Air).

By default, this value is computed automatically based on the component gases in the cylinder. Under normal circumstances the K-factor setting should be set to Automatic. However, if desired, the K-factor can be entered manually by selecting the Manual button. Manual mode turns off automatic computation and allows the K-factor to be typed into the K-factor box.

The K-factor value displayed is relative to NITROGEN. However, the software adjusts the actual K-factor value used relative to the Calibration Reference Gas as specified in the Instrument Calibration screen.

The K-factor value and the type of computation (manual or automatic) is stored as part of the cylinder configuration.

Cylinder Contents

The Cylinder Contents section displays the component gases of the cylinder in table format. The table has an entry for each component gas. Each entry shows the following information for a given component gas.

Interest:	This box is used to indicate the default gas of interest for a multi-component cylinder. The default gas of interest is listed first when selecting component gases in the RUN modes. While there can be more than one gas of interest, only one can be selected as Interest. The gas of interest may be changed at any time from the various RUN modes, regardless of the Interest setting.
Balance:	This box is used to indicate which gas is the BALANCE or CARRIER gas. Only one gas can be the Balance Gas
Gas Name:	The name of the component gas selected from the Gas Library.
Symbol:	The symbol of the selected gas.
Concentration:	Indicates the concentration of the gas in the cylinder. Concentration is shown in units of percentage (%) of the total, or in parts per million (PPM). If the gas is selected as the Balance Gas, its concentration is automatically calculated based on the concentrations of all other gases in the cylinder.
K-Factor:	Gas Correction Factor for the component gas, as computed in the Gas Library screen. This value is computed relative to Nitrogen. This value is for reference only, and is not used directly in the calculation of the Cylinder K-factor.

Managing the Cylinder Library

The Add, Copy and Delete buttons allow you to manage cylinders in the Cylinder library:

Add:	Add a new cylinder to the library.
Copy:	Make a copy of the current cylinder, allowing you to change the name and contents.
Delete:	Deletes the currently selected cylinder from the library.

To add a cylinder to the Cylinder Library

Click *Add* and enter a name for the new cylinder at the prompt. Click the *OK* button. This name will appear in the Name box.

Fill in the desired Information, K-factor and Expiration Date value, if desired

Enter the contents of the cylinder beginning with the balance gas. Using the pull down box under *Gas Name* or *Symbol* in the Contents section, select the desired balance gas. The concentration will be automatically set at 1 million ppm.

Continue adding the remaining component gases by selecting them from the *Gas Name* or *Symbol* pull down box

Select the default gas of interest by clicking the Interest box next to the desired gas.

Once all gases have been added, enter the concentration of each component gas under the concentration box. The units can be changed from ppm to % using the pull down box under units. The balance gas concentration will automatically be filled in as the difference so that all gases add up to 1 million ppm (100%).

As you enter data, the cylinder K-factor will automatically be computed based on the concentration of the individual gases in the cylinder. This total K-factor is displayed next to the cylinder name. If you desire, the total K-factor can be entered manually by clicking on *Manual* in the Name section and then typing the K-factor value in the K-factor box.

When done, click *Close* to save your cylinder and exit the Cylinder Library. The program will display a prompt to allow the new cylinder to be saved. Click the *Yes* button to save.

Alternately, you can click *Add* to add a new cylinder without exiting the Cylinder Library.

To make a copy of a cylinder

1. Select the cylinder to copy using the pull down box under Cylinder Name.
2. Click the *Copy* button to copy the contents section to a new cylinder. At the prompt, enter the name of the new cylinder.
3. Click *OK* button to copy the current contents into the new cylinder.

To delete a cylinder

1. Select the cylinder to delete using the pull down box under Cylinder Name.
2. Click on the *Delete* button to remove the currently selected cylinder.
3. Click on *Yes* to confirm the cylinder deletion.

Editing gases in a cylinder

The following buttons are present on the Cylinder Library form to assist editing the cylinder contents:

Move Up:	Moves the currently selected gas up one position in the table.
Move Down:	Moves the currently selected gas down one position in the table.
Clear:	Deletes the selected gas from the cylinder contents table.
Gas Library:	Brings up the Gas Library form, allowing the addition of new gases while editing the current cylinder.

1. To select a gas to move or delete, click on the name of the gas. The name box will appear red.
2. To move a gas, click on the *Move up* or *Move down* button to move the gas down one position on the list. Continue to click on the *Move down* button until the gas is in the desired location on the list.
3. To delete the selected gas from the cylinder, click the Clear button

Generating Reports

The Report function allows the user to print or export the contents of the cylinders in their cylinder library.
See Section 7 - Generating Reports for further details.

Saving Changes

When all desired Cylinder Library editing is finished, press the *Close* button to return to the main menu window.

5.3 Port Configuration

Port Configuration specifies which gas cylinders are connected to the instrument gas input ports. This information is referenced by all RUN modes of the instrument, and must agree with the actual gas connection to the instrument in order to operate correctly.

To open the Port Configuration screen, select *Configure – Port* from the main menu. You must have a desktop open before you can change the port configuration. If a desktop is not open, then *Configure – Port* will not be accessible.

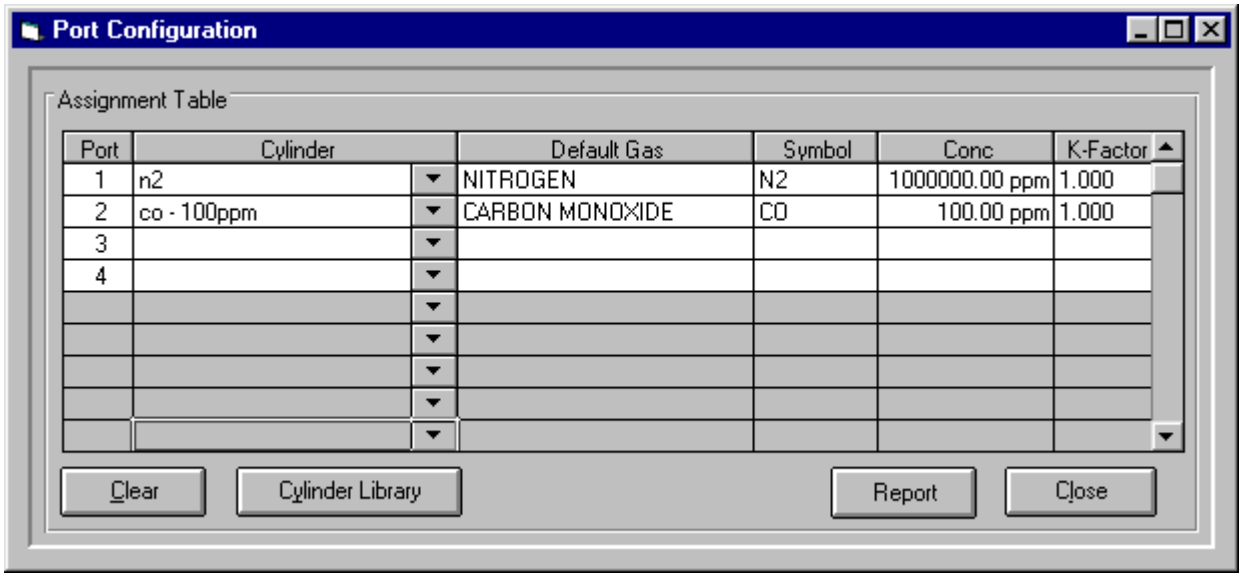


Figure 5 - Port Configuration form

The port configuration in related directly to the currently open Desktop. Only one port configuration can be defined for each desktop.

The cylinder assignments for the port configuration are displayed in the Assignment Table section of the form. A cylinder assignment for a given port consists of the following information:

Port:	Indicates the port number on the rear panel of the instrument.
Cylinder:	Selects the Cylinder assigned to the port. Only one cylinder can be assigned to any port.
Default Gas:	Displays the default gas of interest for the selected cylinder.
Symbol:	Displays the symbol for the default gas of interest.
Concentration:	Displays the concentration of default gas of interest.
K-factor:	Displays the K-factor for the cylinder (relative to Nitrogen).

The following buttons are present on the Port Configuration form to assist editing:

Clear:	Clear the selected entry from the table.
Cylinder Library:	Brings up the Cylinder Library form, allowing the creation of a new cylinder “on the fly”.

To assign a cylinder to a port

The current cylinder assignments for the port configuration are displayed in the port configuration assignment table. To assign a different cylinder to the port, use the pull down box under cylinder, and select the desired cylinder from those listed in the cylinder library.

Note: The cylinder must be listed in the library in order to be assigned to a port.
Click on the cylinder library button to add a new cylinder if required.
See Section 5.2 - Cylinder Configuration for further details.

Once the cylinder is assigned, the default gas, symbol, concentration, and K-factor are automatically placed in the table from the cylinder library. It cannot be edited within the port configuration table. To edit this information, click on *Cylinder Library* and edit from within the library.

To save the cylinder/port assignments, click on *Close*. Click *Yes* to save the changes.

Note: The same cylinder cannot be assigned to more than one port. To use the same cylinder on multiple ports, make a copy of the cylinder with the cylinder library COPY button. See Cylinder Library for further details on copying cylinders.

To Remove a Cylinder from a Port

1. Select the cylinder by clicking on the cylinder name in the assignment table.
2. Click the *Clear* button to remove the cylinder from the port configuration table.

Generating Reports

Reports of the port configuration can be created using the report function. See Section 7 - Generating Reports for further details.

Note: Any changes made to the port configuration will not be reflected in the reports unless you save the Port Configuration first. To save any changes to the Port Configuration, click Close and then Yes.

Saving Changes

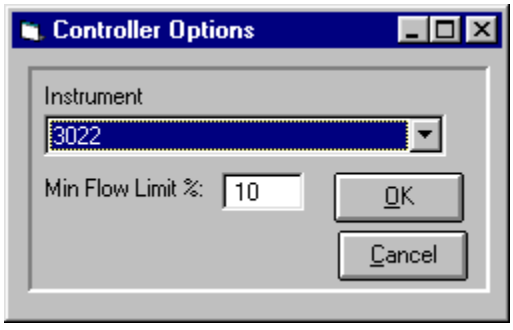
When finished editing port information, click on the *Close* button to return to the main menu window. If any changes have been made, the software will prompt you to save. Click *Yes* to save changes and return to the main menu.

5.4 Instrument Configuration

The Instrument Configuration menu allows for the installation of new instruments and changing the operating parameters. To access the Instrument Configuration options, click *Configure* – *Instrument* and select the desired submenu, which consists of the following items:

Install:	Allows installation of new instruments in the database. Refer to Section 2.3 - Instrument Data Installation
Controller Options:	Opens the controller options form.
Calibrate:	Opens the instrument calibration form. Refer to Section 8.4 - MFC Calibration Procedure for details on calibrating the instrument

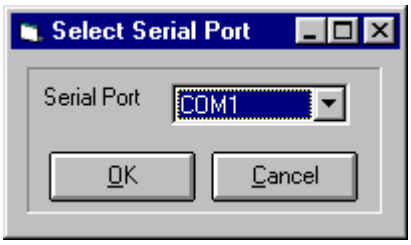
Setting Controller Options



The Controller Options screen allows you to set the Minimum Flow Limit (MFL) of the flow controllers. The MFL is used by the software for automatically selecting the proper flow controller (in a 4040 dilution system), and for generating warning messages when operating the flow controllers below this value. The MFL is expressed as a percentage of the Mass Flow Controller's size, and is normally set to 10%. Changing the value may have an undesirable effect on the operation of the system and should not be done without consulting Environics.

5.5 Serial Port Configuration

The Serial Port configuration screen allows you to select which PC serial port to use for connection to the instrument. Serial Ports COM1-COM4 are supported by the software. To access the Serial Port Configuration screen, click *Configure – Serial Port.* Select the desired serial port from the menu, and then click *OK*.

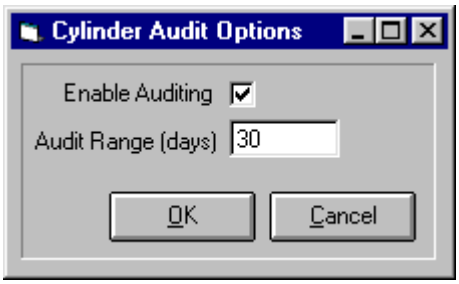


5.6 Cylinder Auditing

The Cylinder Auditing option allows the software to report cylinders that are approaching their expiration date.

In order to use Cylinder Auditing, you must make sure that each cylinder in the cylinder library that you wish to audit has an expiration date assigned. Refer to Section 5.2 - Cylinder Configuration for how to set cylinder expiration dates.

To activate Cylinder Auditing, click *Configure – Cylinder Auditing* from the main menu. Then, check the Enable Auditing box. To receive advanced notice of cylinders about to expire, set the "Audit Range" field to a number greater than zero. This will allow the software to identify cylinders that will expire within the specified number of days.



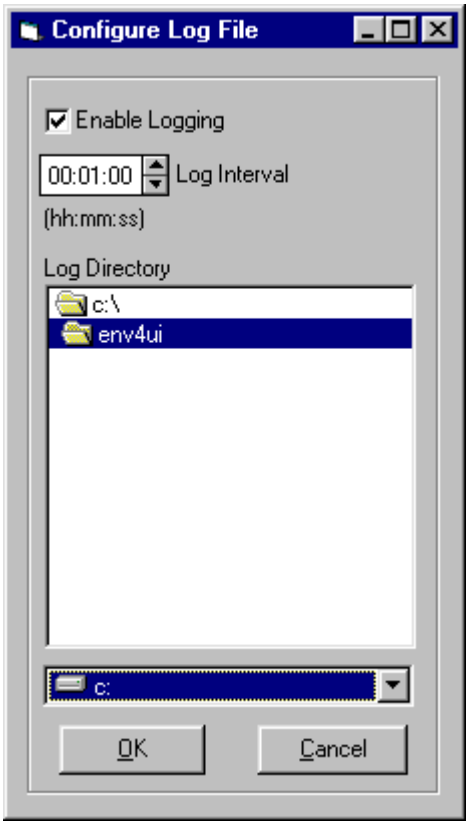
Each time the Environics Series 4000 software is run, it will check the expiration dates on all cylinders. If there are any cylinders that are expired, or are expiring within the Audit Range specified, a warning message is displayed, which gives you the option of viewing the Cylinder Audit report.

Cylinder Audit Report

You can generate a report of all cylinders showing their expiration dates. Select *Reports - Cylinder Audit* from the main menu, and click *Print*. This report will show all cylinders sorted by expiration date, and listing the number of days left before each cylinder expires.

5.7 Log File Configuration

The Log File option is used to generate an audit trail log of all actions performed by the Environics Software. To activate the Log File, click *Configure – Log File*, and then check the Enable Logging box.



The Log Interval setting controls how often the log file is updated. Set this value to the desired update interval. The target and actual flow or concentration values of each gas will be written to the log file at the specified interval.

The Log Directory setting is used to specify where the log files should be stored. Select the desired Directory and Drive by clicking them in the Log Directory window. Click *OK* to save the Log File settings.

Log File Names

The filenames of the log files that are generated follows a fixed format. Log files are named by date, and numbered sequentially where more than one log file was generated on the same date.

The format of the log filename is: YYMMDD.SSS

where YY = year (last 2 digits)
MM = month
DD = date
SSS = sequence (number 001-999)

A new log file is created each time you start a new Run mode (Flow mode, Concentration Mode, etc). While that Run mode is running, data will be written to the log file at the specified interval. Pressing STOP will stop data from being written to the log file. Pressing START again will add new data to the existing log file.

However, selecting a different saved Run mode and then pressing START will cause a new log file to be created, using the next number in the sequence.

For example, suppose we click *Run – Concentration Mode*, and then select the saved Conc mode setting called "CO2 cal 10 ppm" and then press START. Assuming this is the first time we have run Concentration Mode today, the sequence # will be 001. If today's date is Jan 7, 2003, the log file name will be "030107.001". If we press STOP, then press START again, new data will be appended to the existing file.

Now, suppose we press STOP, then select the saved Conc mode setting called "CO2 cal 50 ppm" and press START. Since this is a new Run mode (still Concentration mode, but a different saved setting), a new log file "030107.002" will be created.

Log File Format

A sample of the data stored in the log file is shown below. The exact format of the log file varies, depending on what Run mode is being logged.

```
Environics Series 4000
Instrument:1234
Desktop:project2
Port assignments
Port  1 :n2
Port  2 :co - 100ppm
Port  3 :not assigned
Port  4 :not assigned
Mode:Conc
Name :co - 25ppm
Date,Time,Cylinder,Gas,Target OGC,Actual OGC,Units,Cylinder,Gas,Target OGC,Actual OGC,Units,Cylinder,Gas,Target OGC,Actual
OGC,Units,
   Target Total Flow, Actual Total Flow, Units
11/12/03,11:04:32 AM,n2,NITROGEN,750000.00,0.00,PPM,co - 100ppm,CARBON MONOXIDE,25.00,0.00,PPM,1000.00,0.00,CCM
11/12/03,11:04:33 AM,n2,NITROGEN,750000.00,0.00,PPM,co - 100ppm,CARBON MONOXIDE,25.00,0.00,PPM,1000.00,0.00,CCM
11/12/03,11:04:34 AM,n2,NITROGEN,750000.00,0.00,PPM,co - 100ppm,CARBON MONOXIDE,25.00,0.00,PPM,1000.00,0.00,CCM
```

6. Running the Instrument

The various RUN modes allow the user to operate the instrument and specify the desired gas concentration and flow. There are three main RUN modes: Concentration mode, Flow mode, and Divider mode. The Concentration mode allows the user to specify the desired concentration of gases and total flow rate of the output. The Flow mode allows the user to specify the individual flow rate of each gas. The Divider mode provides an automatic 10 step dilution of a single gas. In addition, the Program and Schedule modes allow the user to group Concentration, Flow and Divider modes together into a set of step-by-step operations (a program) and schedule these operations to run at a specific time and date.

6.1 Concentration Mode

Concentration mode is used to create a blend by entering target gas concentrations for each gas of interest, and the desired total output flow for the mix. The software automatically computes the correct flow for each gas to achieve the desired output. Selecting Concentration from the main Run menu opens the Concentration Mode form as shown in Figure 6. This form allows for both editing stored concentration definitions and for running a selected concentration.

Note: A desktop file must be open to run Concentration mode.

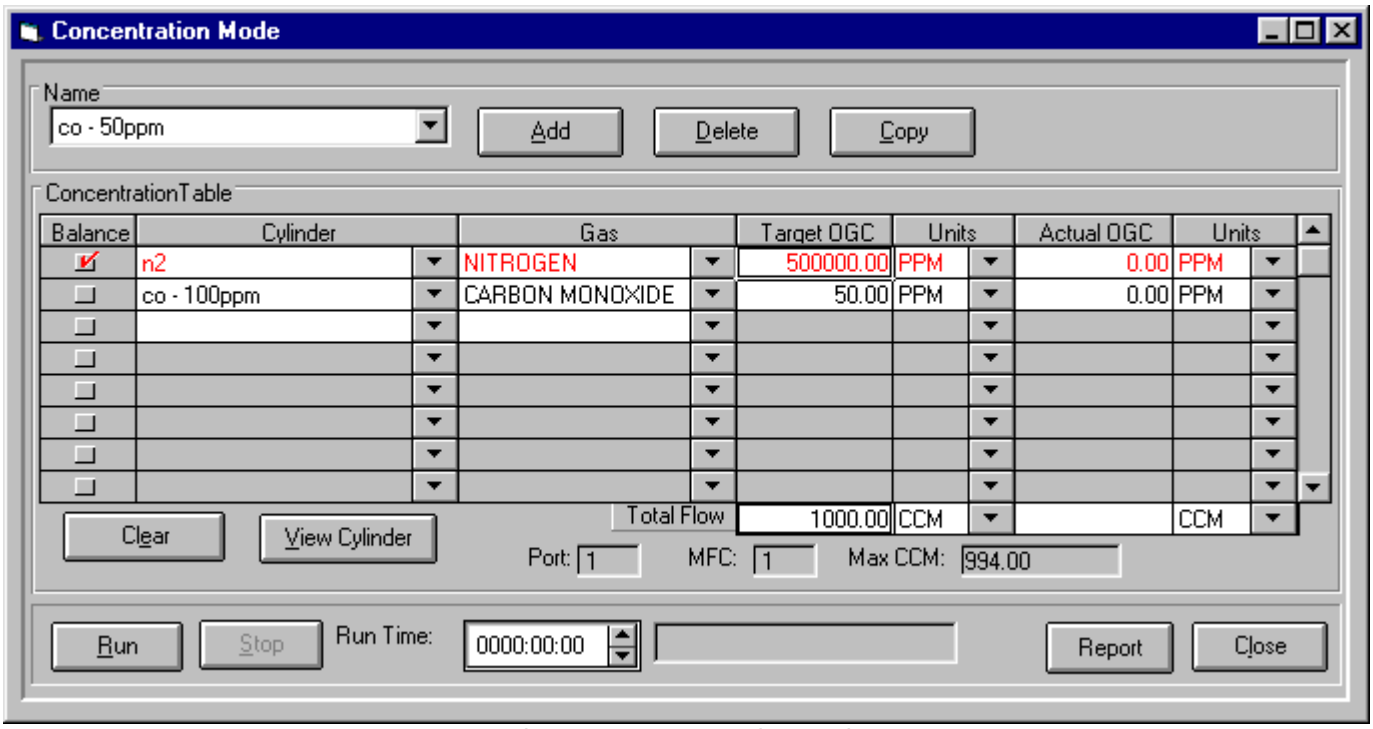


Figure 6 - Concentration Mode

Overview

The system maintains a library of Concentration mode files. Each file in the library is named by the user and contains the associated concentration assignments for the individual cylinders and the total output.

The actual gas input ports and flow controllers used in Concentration mode are automatically selected based on the specified cylinder and output concentrations. Cylinders are mapped to ports based on the port configuration. Concentrations are mapped to controllers based on the physical port to controller connections of the instrument.

Managing Concentration Modes

The Concentration Mode screen displays the current concentration mode file, including its Name and Concentration/Flow settings. You can add, delete or copy these concentration mode files using the three buttons next to the Name box.

Add:	Add a new Concentration Mode file.
Copy:	Copy the current Concentration Mode file, and saving it with a new name.
Delete:	Deletes the current Concentration Mode file.

To Add a Concentration Mode file

1. Click *Add*.
2. Enter the name for the file and click *OK.*
3. A blank Concentration Mode table will appear. Fill in the table with the values you want to use. Refer to Editing Concentration Mode below for details about setting up the concentration mode table.

To Copy a Concentration Mode file

1. Select the file to copy using the pull down box under Name
2. Click the *Copy* button to copy the concentration table to a new file. A prompt will ask for a new name to copy the information to.
3. Enter the new name and click on the *OK* button to copy the current contents into the new file.

To Delete a Concentration Mode file

Select the file to delete using the pull down box under Name.
Click on the *Delete* button to remove the currently selected file.
Click on *Yes* to confirm the file deletion.

Editing the Concentration Mode table

The Concentration Mode table shows the settings for the current concentration mode file. These settings include cylinder and concentration settings and total flow. The items in the table are shown below:

Balance:	Indicates which gas cylinder is to be used as a balance or diluent gas
Cylinder:	The name of the gas cylinder.
Gas:	Indicates the gas of interest for the cylinder. The gas of interest indicates which gas to consider when specifying the Target OGC. This is primarily used with multicomponent cylinders
Target OGC:	This is the desired output gas concentration for this cylinder's specified gas of interest. If the cylinder is checked as the BALANCE cylinder, this is automatically calculated based on the OGC values for specified for the other cylinders.
Actual OGC:	Displays the actual output gas concentration, in the units specified (% or PPM).
Units:	Selects the OGC units as either % or PPM.
Total Flow:	Used to specify the total flow rate of the output.

Set up the concentration mode table as follows:

1. Specify each desired source cylinder by using the pull down box in the concentration table under Cylinder. Select the cylinder by clicking on it. The port is automatically selected based on the current cylinder/port assignments. The port number is displayed in the Port box.

   Note: The only available cylinders listed are those assigned to ports. If the cylinder you want is not listed, you must modify your port settings. See Port Configuration for further details.

2. Next, specify the gas of interest by using the pull down box under Gas next to the cylinder name. Select the component gas of interest by clicking on it. If you want to see the contents of the cylinder click on *View Cylinder*.

3. Once all of the cylinders are entered, select the balance gas by clicking in the balance box beside the cylinder name.
4. The next step is to specify the desired output concentration for each cylinder. Type the value in the Target OGC (output gas concentration) box. If desired, the units can be changed between ppm and % by using the pull down box under units.

Note: The Target OGC for the balance gas is automatically calculated and cannot be modified.

5. Finally, enter the Total Flow is entered by typing in its value. The units can be changed from CCM to LPM by using the pull down menu under units. The value entered is used by the system to compute the target concentration of the balance gas and the required flow of each controller.

When entering data, the system will check that the values entered are allowable, based on several factors, including Cylinder Gas Concentration, Total Flow and the range of the Mass Flow Controllers. The software will generate a warning and attempt to correct the entry so that it is an acceptable value.

The actual ports and controllers used in this Concentration mode are automatically selected based on the specified cylinder, gas concentrations and total flow. Clicking on a particular cylinder in the table will display information about the Port and Mass Flow Controller (MFC) selected by the software. The maximum flow for that flow controller is displayed as well.

To Remove a cylinder from the concentration table

1. Select the cylinder by clicking on the cylinder name in the table.
2. Click the *Clear* button to remove the cylinder from the concentration table.

To View the contents of a cylinder

Select the cylinder by clicking on the cylinder name in the table.
To view the contents of a cylinder

Click the *View Cylinder* button to view the cylinder in the Cylinder Library.
-or-
Click and hold the right mouse button to show the Cylinder Component OGC window (Figure 7 below). This displays detailed information about the cylinder contents, the calculated Output Gas Concentration (OGC) for each gas component, and the flow rate of the gas for that cylinder. Release the right mouse button to close the window.

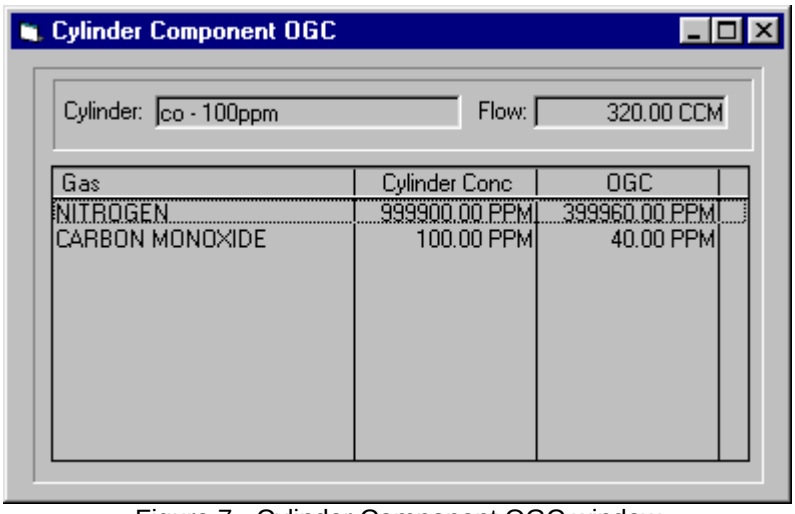


Figure 7 - Cylinder Component OGC window

Running Concentration Mode

The Run Control section located at the bottom of the screen is used to run the specified gas concentrations. The instrument can be run either manually or with a timer. Once the instrument is running, all controls in the Concentration Mode form are disabled except for cylinder selection, target concentration value, and total flow value.

Running Manually

1. To start flowing gases, click the *Run* button. The software will display in real time the actual gas concentrations and flow rate. The run status box will indicate the elapsed run time.
2. To stop the flow, click *Stop*.

Running with a Timer

To flow gases for a specific time duration, set the desired run time in the Run Time box. You can enter the value directly, or use the up/down arrows to set the time.

Click the *Run* button. The instrument will run for the specified length of time and then stop. While the instrument is running, the run status box indicates the elapsed run time. The software will display in real time the actual gas concentrations and flow rate.

Changing settings while running

To change the cylinder selection while the instrument is running

1. Use the pull down box to change the current cylinder to the desired one.
2. Press *Enter* to set the new cylinder.

To change the target concentration while the instrument is running

Click on the Target OGC box for the desired cylinder.

Enter the new target value and press *Enter*. This sets the new cylinder output concentration and recalculates the balance concentration to maintain the desired output flow.

To change the total flow while the instrument is running

1. Click on the Total Flow box.
2. Enter the new total flow value and press *Enter*. This sets the new value and recalculates the balance concentration to produce the new flow.

To view the total flow for each cylinder

Click and hold the right mouse button.

The Actual OGC column will change to Actual Flow, and the actual flow rate of each cylinder will be displayed. This will also display the Cylinder Component OGC window. Release the right mouse button to return the display to normal.

Generating Reports

The Report button allows the user to print or export the contents of the concentration table.
See Section 7 - Generating Reports for further details.

Saving Changes

When finished running Concentration Mode, click the *Close* button to return to the main menu window. If any changes have been made, the software will prompt you to save. Click *Yes* to save changes and return to the main menu.

6.2 Flow Mode

Flow mode allows direct control of the flow rate of each gas. This mode is an alternative to Concentration mode. In flow mode, the output flow rate of each gas is set directly by the user. Selecting *Flow* from the main *Run* menu opens the Flow Mode form as shown in Figure 8. This form allows for both editing stored flow definitions, and for running the selected flow.

The actual ports and controllers used in this mode are automatically selected based on the specified cylinder and output flows. Cylinders are mapped to ports based on the current port configuration. Flows are mapped to controllers based on the physical port connections of the instrument.

Note: A desktop file must be open to run the Flow mode.

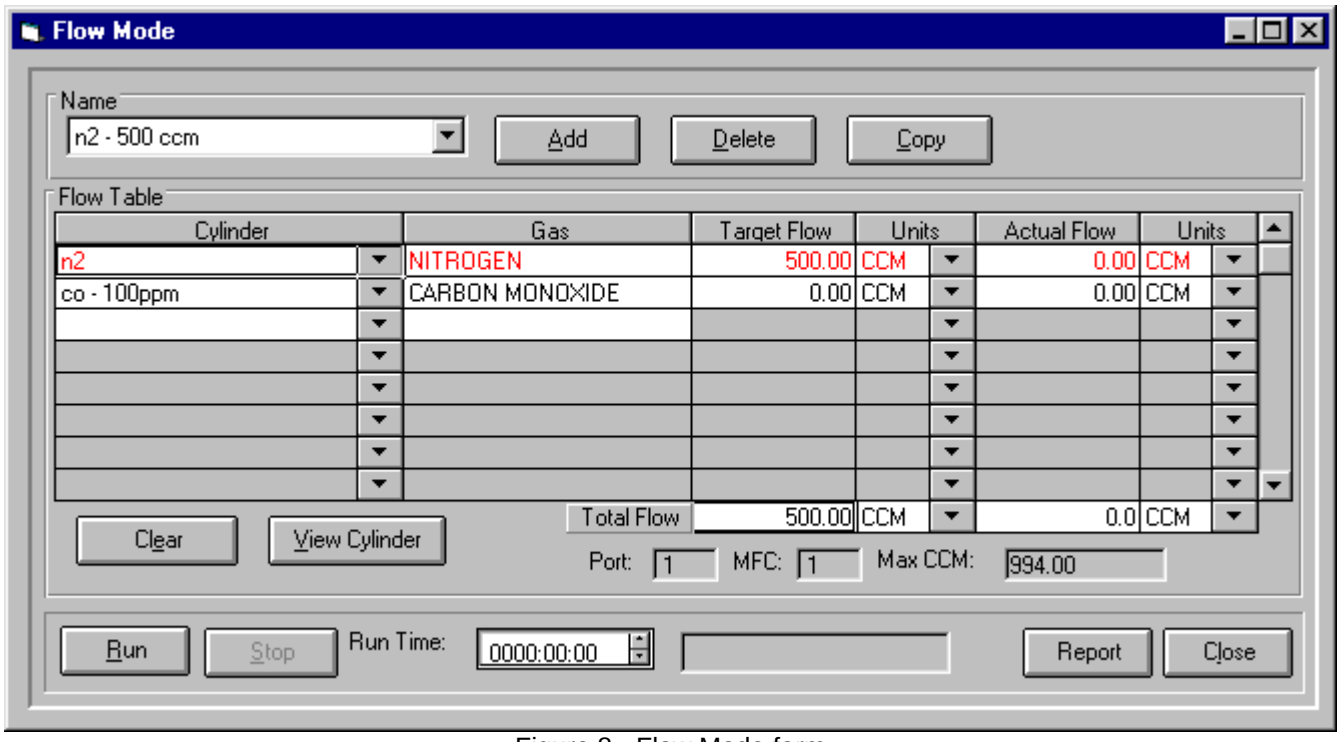


Figure 8 - Flow Mode form

Overview

The system maintains a library of Flow Mode file. Each file in the library is named by the user and contains the associated flow assignments of the individual cylinders. Only one stored Flow definition is active at a time. The currently selected configuration is displayed in the pull down list box. A different configuration is selected from the library by clicking the pull down box and selecting the desired configuration.

Managing Flow Modes

The Flow Mode screen displays the current flow mode file, including its Name and Flow settings. You can add, delete or copy these flow mode files using the three buttons next to the Name box.

Add:	Add a new Flow Mode file.
Copy:	Copy the current Flow Mode file, and saving it with a new name.
Delete:	Deletes the current Flow Mode file.

To Add a Flow Mode file

1. Click *Add*, then enter the name for the file and click *OK*.
2. A blank Flow Mode table will appear. Fill in the table with the values you want to use. Refer to Editing Flow Mode below for details about setting up the Flow mode table.

To Copy a Flow Mode file

Select the file to copy using the pull down box under Name
Click the *Copy* button to copy the Flow table to a new file. A prompt will ask for a new name to copy the information to.
Enter the new name and click the *OK* button to copy the current contents into the new file.

To Delete a Flow Mode file

1. Select the file to delete using the pull down box under Name.
2. Click the *Delete* button to remove the currently selected file.
3. Click on *Yes* to confirm the file deletion.

Editing the Flow Mode table

The Flow Mode table shows the settings for the current flow mode file. These settings include cylinder, gas and target flow. The items in the table are shown below:

Cylinder:	Selects the desired cylinder to use.
Gas:	Displays the default gas of interest for the selected cylinder.
Target Flow:	Used to enter the desired flow rate for this cylinder.
Actual Flow:	Displays the actual flow rate for this cylinder. This column only displays values when the instrument is actually running the specified flow.
Units:	Selects the units for entry/display of flow rates.
Total Flow:	Displays the total Target and Actual flow rates for the output. These values are the sum of the cylinder flows in their respective columns.

Set up the flow mode table as follows:

1. Specify each source cylinder by using the pull down box in the flow table under Cylinder. Select the cylinder by clicking on it. The port for the flow is automatically selected based on the selected cylinder/port assignments. The port number is displayed in the Port box.

Note: The only available cylinders listed are those assigned to ports. If the cylinder you want is not listed, you must modify your port settings. See Port Configuration for further details.

2. The gas of interest is automatically selected based on the default gas of interest originally specified for the cylinder. Since the system is not controlling the gas concentration, the gas of interest shown is irrelevant. Only the total gas flow of the cylinder is controlled. However, if you want to have a different gas displayed, click the *View Cylinder* button to change the gas of interest for the cylinder in the Cylinder Library.
3. Once all of the cylinders are entered, the next step is to specify the desired flow rate for each cylinder. Type the value in the Target Flow box. The maximum flow for the selected cylinder is shown in the Max CCM box. If desired, the units can be changed between CCM and LPM by using the pull down box under units.

Note: The Total Flow value is automatically calculated and cannot be modified.

When entering data, a warning will appear if the Target Flow is greater than the maximum flow allowed for that flow controller. The warning will provide the maximum CCM value allowed, and automatically enter that value in the Target Flow field.

The system automatically selects the required gas ports and flow controllers based on the specified cylinders and flow rates. Clicking on a particular cylinder in the table will display information about the Port and Mass Flow Controller (MFC) selected by the software. The maximum flow for that flow controller is displayed as well.

To Remove a Cylinder from the Flow Table

1. Select the cylinder to clear by clicking on the cylinder name in the table.
2. Click the *Clear* button to remove the cylinder from the flow table.

To View the contents of a cylinder

Select the cylinder by clicking on the cylinder name in the table.
To view the contents of a cylinder, click the *View Cylinder* button.

Running Flow Mode

The Run Control section located at the bottom of the form is used to run the specified gas flows. The instrument can be run either manually or with a timer. Once the instrument is running, all controls in the Flow Mode form are disabled except for cylinder selection and target flow rate.

Running Manually

1. To start flowing gases, click on the *Run* button. The software will display in real time the actual flow rates of the gases. The run status box will indicate the elapsed run time.
2. To stop the flow, click *Stop*.

Running with a Timer

To flow gases for a specific time duration, set the desired run time in the Run Time box. You can enter the value directly, or use the up/down arrows to set the time.
Click the *Run* button. The instrument will run for the specified length of time and then stop. While the instrument is running, the run status box indicates the elapsed run time. The software will display in real time the actual flow rates of the gases.

Changing settings while running

To change the cylinder selection while the instrument is running

1. Use the pull down box to change the current cylinder to the desired one. You cannot
2. Press *Enter* to set the new cylinder.

To change the target flow value while the instrument is running

Click on the Target Flow box for the desired cylinder.
Enter the new target value and press *Enter*. This sets the new cylinder flow value and recalculates the total flow. Due to the change in flow, the flow controller selection may change.

Generating Reports

The Report button allows the user to print or export the contents of the flow table.
See Section 7 - Generating Reports for further details.

Saving Changes

When finished running Flow Mode, click the *Close* button to return to the main menu window. If any changes have been made, the software will prompt you to save. Click *Yes* to save any changes and return to the main menu.

6.3 Divider Mode

Divider mode provides an automated multiple step dilution of gas. The dilution mixes two gases together with the ability to vary the concentration of the mix over the specified run time. The two gases are referred to as the Span gas and the Dilution Gas. The Span gas is the gas that you want to divider. The Dilution gas is used to dilute the span gas to the specified output concentration. The dilution is varied over time by setting the desired concentrations of the span gas for a series of time steps. Typically the concentration will change at each step. The dilution is accomplished by mixing the span gas with the dilution gas. Selecting *Divider* from the main *Run* menu opens the Divider Mode form as shown in Figure 9. This form allows for both editing of stored divider configurations, and for running a selected configuration.

The actual ports and controllers used in this mode are automatically selected based on the specified cylinder and output concentrations. Cylinders are mapped to ports based on the currently active Port library. Concentrations are mapped to controllers based on the physical port connections of the instrument.

Note: A desktop file must be open to run the Divider mode.

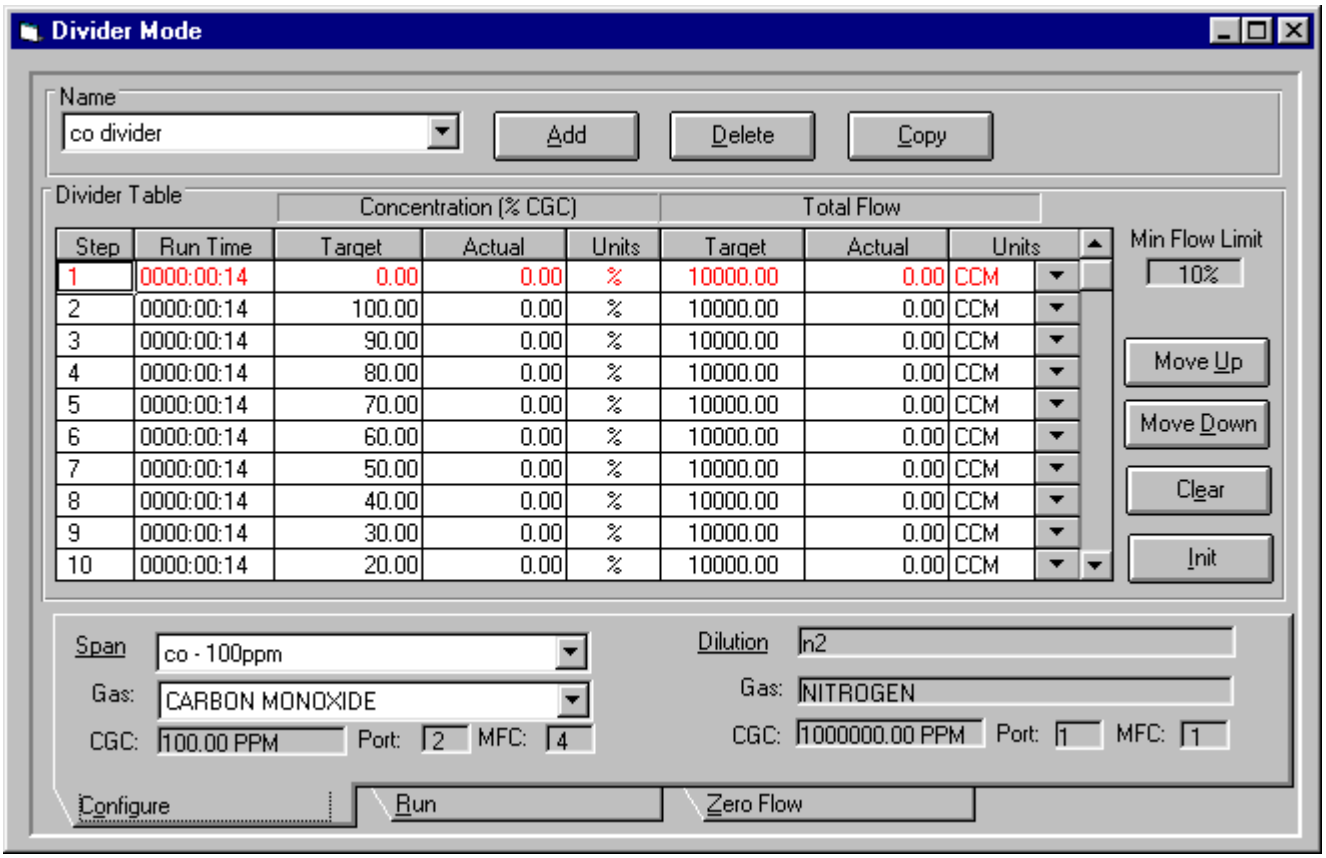


Figure 9 - Divider Mode form

Overview

The system maintains a library of Divider Mode files. Each file in the library is given a name by the user and contains the associated divider step assignments. Only one stored divider configuration is active at a time. The currently selected file is displayed in the pull down list box. A different file is selected from the library by clicking the pull down box and selecting it from the list of available files.

Since there are a large number of buttons and data entry fields on the Divider Mode table, it is split into 3 control tabs. These tabs appear at the bottom of the Divider Mode screen and are labeled *Configure*, *Run* and *Zero Flow.* Each tab will be discussed in more detail below.

Managing Divider Modes

The Divider Mode screen displays the current divider mode file, including its Name and all Divider settings. You can add, delete or copy these divider mode files using the three buttons next to the Name box.

Add:	Add a new Divider Mode file.
Copy:	Copy the current Divider Mode file, and saving it with a new name.
Delete:	Deletes the current Divider Mode file.

To Add a Divider Mode file

1. Select *Add*, enter a name for the file and click *OK.*
2. A blank Divider Mode table will appear. Fill in the table with the values you want to use. Refer to Editing Divider Mode for details about setting up the Divider Mode table.

To Copy a Divider Mode file

1. Select the file to copy using the pull down box under Name.
2. Click the *Copy* button to copy the divider table to a new file. A prompt will ask for a new name to copy the information to.
3. Enter the new name and click the *OK* button to copy the current contents into the new file.

To Delete a Divider Mode from the Library

1. Select the file to delete using the pull down box next to the Name box.
2. Click the *Delete* button. Click the *Yes* button to confirm the deletion.

Editing Divider Tables

The Divider Mode table shows the settings for the current divider mode file. The Divider Table contains 11 steps, with each step consisting of the following information:

Step:	The step number in the divider table. There are 11 steps available. The first step is normally used to perform a "Zero" operation (where only dilution gas is used to establish a zero reading on an analyzer)
Run Time:	The amount of time the step runs (in hours:minutes:seconds).
Concentration (%CGC):	The desired percentage of Span gas in the final mixture.
Target Concentration:	The desired output concentration (as a % of the span cylinder concentration).
Actual Concentration:	The actual output concentration from the instrument. Values are displayed only when the step is running.
Total Target Flow:	The desired output flow of the step.
Actual Flow:	The actual output from the instrument. This column only displays values when the instrument is actually running the specified step.
Units	Flow units are selectable between CCM and LPM
Min Flow Limit:	This value is displayed on the right side of the screen and indicates the level of flow at which the instrument automatically switches from one flow controller to another. This value is set in the *Configure–Instrument–Controller Options* menu.

The *Configure* tab section also displays the following information.

Span Cylinder:	Selects which cylinder to use as the span gas. The available selections depend on the cylinders listed in the PORT CONFIGURATION screen. The Span cylinder is set by pulling down the list and selecting the desired cylinder.
Span Gas:	Selects the gas of interest in the Span cylinder. Set the gas of interest by pulling down the list box and selecting the desired gas.
Span Gas CGC:	Displays the concentration of the gas of interest in the span cylinder.

Span Port:	Displays the Port Number that the span cylinder is connected to. Its value depends on the cylinders in the Port Configuration screen.
Span MFC:	Displays the Mass Flow controller to be used for the span gas in a particular step. Click on the desired Step # to see the MFC to be used.
Dilution Cylinder:	Displays the cylinder connected to the dilution port. This is specified in the PORT CONFIGURATION screen and corresponds to the cylinder connected to port 1.
Dilution Gas:	Displays the gas of interest in the dilution cylinder.
Dilution Gas CGC:	Displays the concentration of the gas of interest in the cylinder.
Dilution Port:	This is fixed to port 1, which is reserved as the Dilution port.
Dilution MFC:	Displays the Mass Flow controller to be used for the dilution gas in a particular step. Click on the desired Step # to see the MFC to be used.

To edit the divider mode table, click the *Configure* tab at the bottom of the divider screen.

1. Specify the span gas cylinder and the gas of interest using the pull down boxes beside Span and Gas. The port and CGC will automatically be selected.

Note: The cylinder assigned to port 1 is automatically selected as the diluent. To modify the dilution cylinder, you must modify the port configuration. See Section 5.3 - Port Configuration for further details.

For each divider step, enter the following:

2. Specify the desired run time for the step by clicking in the run time box and typing in the desired value. Time is specified in Hours:Minutes:Seconds format. The up and down arrows can also be used to set the run time. Press *Enter* to accept the entered time.
3. Specify the desired concentration of the Span gas by clicking on the Target concentration box and typing in the desired value. Press *Enter* to accept the value entered. The concentration value is expressed as a percent of the Cylinder Gas Concentration of the span gas in the span cylinder. For example, if the Span Cylinder contains 10 ppm CO2, entering a value of 90% will generate 9 ppm of CO2 at that step. The Span CGC of the gas is displayed in the *Configure* tab for your reference.
4. Specify the desired total flow rate. Click on the Target Total Output Flow box and type in the desired value. Press *Enter* to accept the value entered.

Note: A warning will appear if the Target Flow is out of range. The program will automatically adjust the value to the maximum flow rate.

Once all of the steps have been entered, the mode is ready to be run.

Initializing the Divider Table

The *Init* Button provides a convenient way to quickly initialize all steps of the divider mode table. Initializing the table sets the flow rate for each step to the maximum allowable flow and sets the CGC for each step to 10% increments.

1. Click the *Init* button. This will bring up the Initialize Divider Table window. Enter the run time to be set for all steps by clicking on the run time box and entering the value or using the up/down arrows. The target total flow is preset to the maximum allowable flow rate and cannot be changed from here (it can be manually changed by editing each step individually).
2. Click *OK* to initialize the divider mode table. If individual changes are desired for a specific step, click on the box to change and type in the corrected value. Click *Enter* to set the value in the table.

To Remove a Step from the Divider Table

1. Select the step to clear by clicking on the step number in the table.
2. Click the *Clear* button to remove the step from the divider table.

To Move a Step within a Divider Table

1. To move a step up or down in the list, click on the step number. The step will be highlighted in red.
2. To move the step down, click the *Move down* button. To move the step up, click the *Move Up* button. Continue to click *Move up/down* until the step is in the desired location on the list.

Running Divider Mode

The Divider Mode can be run automatically or manually. In automatic mode, the instrument starts at the first step and automatically runs through all steps for the specified run times. In manual mode, each step is started and stopped manually, and steps can be run in any order.

To run divider mode, click the *Run* tab at the bottom of the divider screen.

Running in Automatic mode

In automatic mode, the instrument starts at the lowest step and automatically run through all steps for the specified run times. The run time counter shows the total elapsed time. The instrument stops when all steps are executed or when the *Stop* button is pressed.

1. To start, click the *Auto* button. The instrument will start at the lowest step and automatically run through all steps for the specified run times. The run time counter shows the total elapsed time.
2. To stop the current step and advance to the next, click the *Advance* button.
3. The instrument will stop when all steps are executed. To stop the instrument before the mode is complete, click the *Stop* button.

Running in Manual Mode

In manual mode, each step is run individually and runs until it is manually stopped. Since each step is run individually, they can be run in any order.

Select the desired step by clicking on it.
Click the *Manual* button. The instrument will start running and continue until the *Stop* button is pressed. While the instrument is running, the run status box indicates the elapsed run time.
To stop the step from running, click the *Stop* button.

Running a "Zero Flow"

The Manual Mode can be used to run a "Zero" step (dilution gas only), but this can be done more easily using the Zero tab. Click the *Zero* tab at the bottom of the divider screen, enter the desired flow rate, and click *Run*. When finished, click the *Stop* button.

Generating Reports

The Report button allows the user to print or export the contents of the divider table.
See Section 7 - Generating Reports for further details.

Saving Changes

When finished running Divider Mode, click the *Close* button to return to the main menu window (click the Run tab first if the Close button is not visible). If any changes have been made, the software will prompt you to save. Click *Yes* to save any changes and return to the main menu.

6.4 Program Mode

Program mode allows the three basic run modes (Concentration, Flow and Divider) to be grouped together into a "Program", or sequence of operations. Each step in the program is executed in sequence until all steps have been completed. Selecting *Program* from the main *Run* menu opens the Program Mode form as shown in Figure 10. This form allows Programs to be edited, stored, and run.

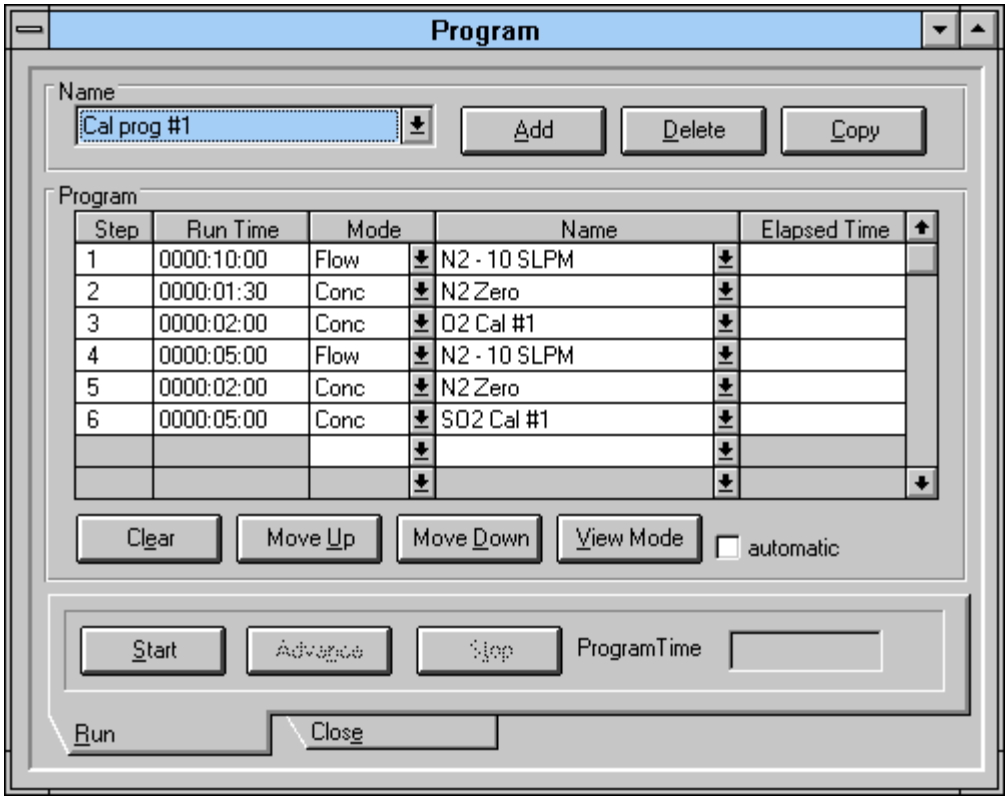


Figure 10 - Program Mode form

Managing Program Mode

The system maintains a library of Program Mode files. Each file in the library is given a name by the user and contains the associated program step assignments. Only one stored program file is active at a time. The currently selected file is displayed in the pull down list box. A different file is selected from the library by clicking the pull down box and selecting it from the list of available files.

The Program Mode screen displays the current program mode file, including its Name and all settings. You can add, delete or copy these files using the three buttons next to the Name box.

Add:	Add a new Program Mode file.
Copy:	Copy the current Program Mode file, and saving it with a new name.
Delete:	Deletes the current Program Mode file.

To Add a Program Mode file

1. Click *Add,* enter a name for the file and click *OK*.
2. A blank Program Mode table will appear. Fill in the table with the values you want to use. Refer to Editing Program Mode below for details about setting up the Program Mode table.

To Copy a Program Mode file

1. Select the file to copy using the pull down box under Name.
2. Click the *Copy* button to copy the Program table to a new file. A prompt will ask for a new file name to copy the information to.
3. Enter the new name and click the *OK* button to copy the current contents into the new file.

To Delete a Program Mode file

Select the file to delete using the pull down box next to the Name box.
Click the *Delete* button. Click the *Yes* button to confirm the deletion.

Editing Programs

Once a program is selected, its steps are displayed in the Program section of the form. A program step consists of the following information:

Step:	The step number of the program.
Run Time:	The amount of time to run this step.
Mode:	The run mode for the step (FLOW, CONC, or DIVIDER).
Name:	The name of the run mode.
Elapsed Time:	Displays how long the current step has been running.

To edit the program mode table, perform the following steps:

1. Click the Mode pull down box in the row of the desired step. Select the desired run mode (Conc, Flow, or Divider).
2. The available saved mode names will appear in the Name box. Using the pull down box under Name, select the desired run mode file.
3. Click in the Run Time box to set the run time for the mode (HH:MM:SS). The Run Time for a divider mode cannot be set here, since the total run time is set within the particular Divider Mode file.
4. Repeat until all desired steps have been entered.

To Move a Step within a Program Table

To move a step up or down in the list, click on the step number. The step will be highlighted in red.
To move the step down, click on the *Move down* button. To move the step up, click on the *Move Up* button. Continue to click *Move up/down* until the step is in the desired location on the list.

To Remove a Step from the Program Table

1. Select the step to clear by clicking on the step number in the table.
2. Click the *Clear* button to remove the step from the program table.

To View/Edit the Mode settings for a step

Click on the step number of the mode you wish to view.
Click the *View Mode* button to view the details of the step. If desired, you can make changes to the displayed mode simply by changing the desired values. When finished viewing/editing, click the *Close* button to return to the Program Mode screen.

Running Programs

To run program mode, click the *Run* tab at the bottom of the screen. The RUN tab contains buttons to control the current program. These buttons are:

Start:	Start the current program running at step 1.
Advance:	Abort the current program step and advance to the next step.
Stop:	Stop the current program.

To Run a Program Mode

Press the *Start* button to begin the current program from step 1. While the program is running, the Elapsed time box indicates the elapsed time for the current step. In addition, the Program Time box indicates the total elapsed time.
To abort the current step and proceed to the next, press the *Advance* button.
To stop the program before completion, click the *Stop* button.

Viewing a Mode while the program is running

Normally, when a program is running, the currently running step is only highlighted. If desired, you can press the *View Mode* button to display the running step in its native Run Mode window.

Alternately, checking the *AutoView* checkbox will cause the currently running step to be viewed in its native Run Mode window, just as if the *View Mode* button was pressed. The Run Mode window will automatically pop up as each step in the program is run.

Generating Reports

The *Report* button allows the user to print or export the contents of the program mode table.
See Section 7 - Generating Reports for further details. The *Report* button is located on the *Close* tab. Click the *Close* tab at the bottom of the screen to access this button.

Saving Changes

When finished running Program Mode, click the *Close* button to return to the main menu window (click the Close tab first if the *Close* button is not visible). If any changes have been made, the software will prompt you to save. Click *Yes* to save any changes and return to the main menu.

6.5 Schedule Mode

Schedule mode allows all other run modes (CONCENTRATION, FLOW, DIVIDER and PROGRAM) to be scheduled to run at a specific time and date. Items can be scheduled as one-time events, or may be recurring events. A schedule may contain an unlimited number of items, and the software allows storage and recall of multiple schedules. Selecting *Schedule* from the main *Run* menu opens the Schedule Mode form as shown in Figure 11. This form allows Schedules to be edited, stored, and run.

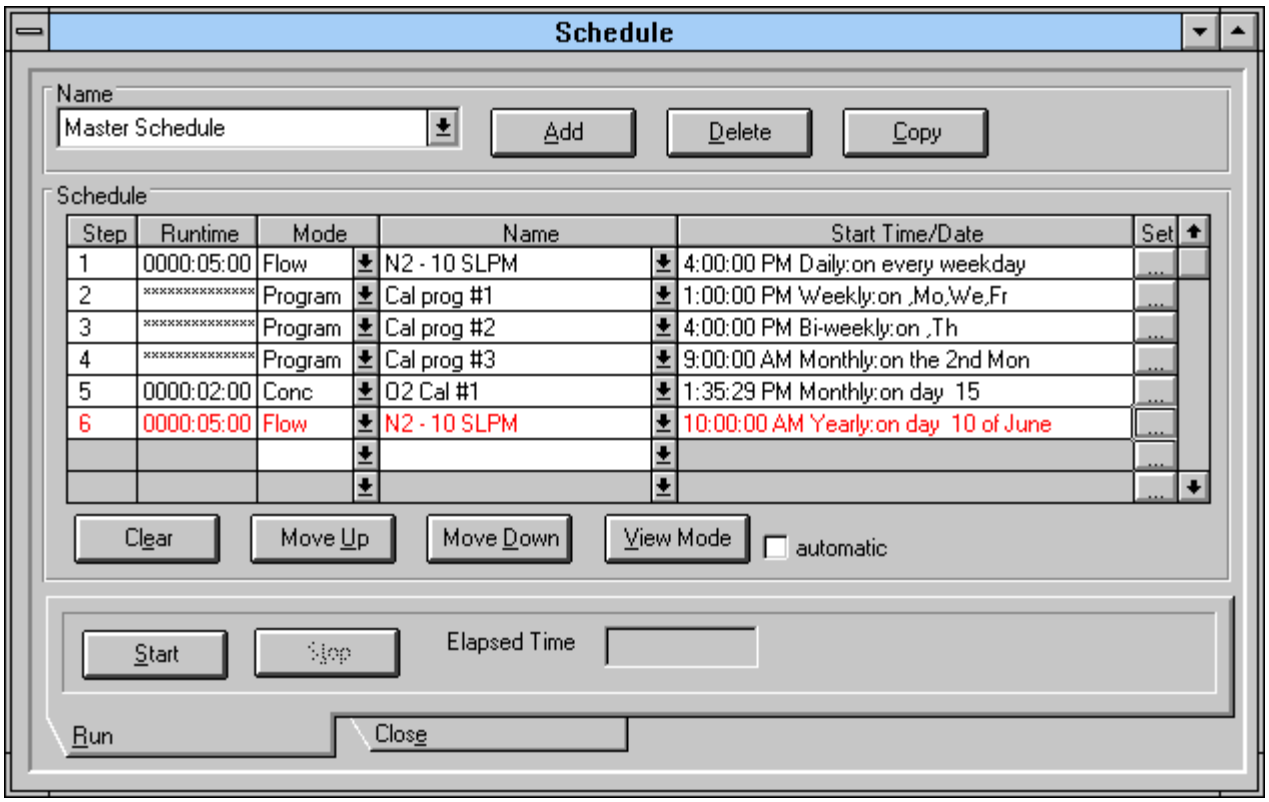


Figure 11 - Schedule Mode form

Editing Schedules

The system maintains a library of stored schedules, which are named by the user. Each file in the library is given a name by the user and contains the associated schedule items. Only one stored schedule file is active at a time. The currently selected file is displayed in the pull down list box. A different file is selected from the library by clicking the pull down box and selecting it from the list of available files.

The Schedule Mode screen displays the current schedule mode file, including its Name and all settings. You can add, delete or copy these files using the three buttons next to the Name box.

Add: Adds a new Schedule to the library.
Copy: Creates a new Schedule and copies data from the current schedule table.
Delete: Deletes the currently selected schedule from the library.

To Add a new Schedule

1. Click *Add,* enter a name for the file and click *OK*.
2. A blank Schedule Mode table will appear. Fill in the table with the values you want to use. Refer to Editing Schedule Mode below for details about setting up the Schedule Mode table.

To Copy a Schedule to a new file

Select the mode to copy using the pull down box under Name.
Click the *Copy* button to copy the schedule table to a new file. A prompt will ask for a new mode name to copy the information to.
Enter the new name and click the *OK* button to copy the current contents into the new file.

To Delete a Schedule

1. Select the mode to delete using the pull down box next to the Name box.
2. Click the *Delete* button. Click the *Yes* button to confirm the deletion.

Editing Schedules

Once a schedule is selected, its items are displayed in the Schedule section of the form. A schedule item consists of the following information:

Item:	The item number of the schedule.
Mode:	The run mode for the schedule item (FLOW, CONC, DIVIDER, or PROGRAM).
Name:	The name of the run mode.
Run Time:	The amount of time this item will be run. This must be specified for CONC and FLOW mode only. For DIVIDER and PROGRAM mode items, the runtime is controlled by the total time required to run the DIVIDER/PROGRAM item, and cannot be changed from here.
Start Time/Date:	Displays the scheduled start time/date of the item.
Set:	Button used to enter the start time/date editor.

To edit the schedule mode table, perform the following steps:

Click the Mode pull down box in the row of the desired item. Select the desired run mode (Conc, Flow, Divider, or Program).
The available saved mode names will appear in the Name box. Using the pull down box under Name, select the desired run mode file.
Click in the Run Time box to set the run time for the mode (HH:MM:SS) – this applies to Flow and Conc mode only.
Click the SET button to set the time/date for this item. Refer to Setting the Start Time/Date below for details.
Repeat until all items have been scheduled.

To Remove an Item from the Schedule

Select the item to clear by clicking on the item number in the table.
Click the *Clear* button to remove the item from the Schedule table.

To Move an item within a Schedule

To move an item up or down in the list, click on the item number. The item will be highlighted red.
To move the item down, click on the *Move down* button. To move the item up, click on the *Move Up* button. Continue to click *Move up/down* until the item is in the desired location on the list.

Note: The order of items in a schedule does not affect the order in which items are run. Items are always run based on the time/date specified.

To View/Edit the Mode settings for an item

1. Click on the item number of the mode you wish to view.
2. Click the *View Mode* button to view the details of the item. If desired, you can make changes to the displayed mode simply by changing the desired values. When finished viewing/editing, click the *Close* button to return to the Schedule Mode screen.

Setting the Start Time/Date

Clicking the Set button will open the Time/Date editor. This is where the Start time and date of the item is set. The Time/Date schedule can be set as a one time or a recurring event.

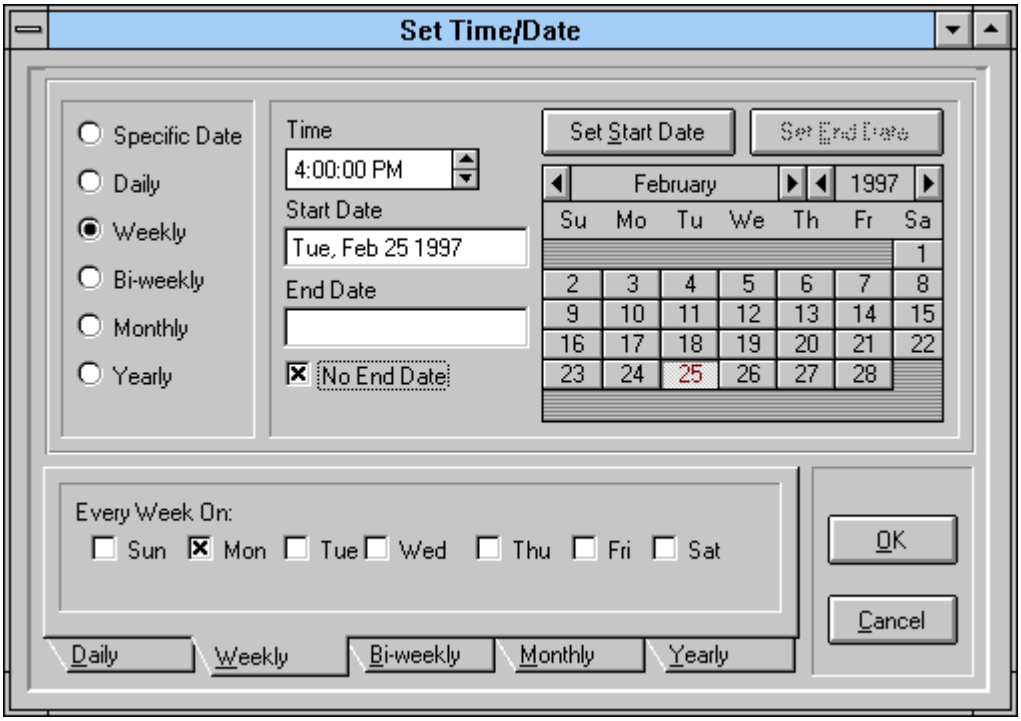


Figure 12 - Schedule Time/Date editor

To set the Time/Date:

1. For the desired item, click the *Set* button to access the Time/Date editor
2. Select the type of event either using buttons on the left side of the screen. For an item that is to occur only once, click the *Specific Date* button. For a repeating event (such as Daily, Weekly, etc.), click the button for the desired repeat frequency.
3. For a repeating event, fill out the appropriate options at the bottom of the screen. The options displayed will vary, depending on the type of repeat chosen. If an item is repeated weekly or biweekly, select the day(s) of the week that it is to be run by clicking in the appropriate square(s). For items repeated monthly, select the day of the month on which to repeat the item (such as every first of the month or every third Monday). For items repeated yearly, select the day and month on which to repeat the item (such as every January 1 or every third Monday in May).
4. Enter the desired start time, by clicking on the Start Time box and typing in the time, or by clicking the up/down arrows.
5. Select the desired start date from the calendar by clicking on the day and then clicking the *SET START DATE* button. Use the left/right buttons to change the month or year displayed. For a Specific Date event, this is the date that the item will be run. For repeating events, this is the date that the event will start to repeat.
6. For repeating events, select the desired end date from the calendar by clicking on the day and then clicking the *SET END DATE* button. This allows the repeating event to occur within a specific range of dates. Alternately, if this item is to be repeated indefinitely, click the *NO END DATE* box. End Date is not used for a Specific Date event.

Running Schedules

To run a schedule, click the *Run* tab at the bottom of the screen. The RUN tab contains buttons to control the current schedule. These buttons are:

Start:	Start the current schedule running.
Stop:	Stop the current schedule.

To Run in Schedule Mode

1. When the *Start* button is pressed, the scheduler evaluates all items in the schedule to see if any are ready to run. The schedule is reevaluated at midnight of each day to determine if any items are to be run during the day. The status of each item is indicated by the color of its *Set* button.

RED:	Item has not been run/is not scheduled to run today.
YELLOW:	Item is scheduled to be run today.
GREEN:	Item is currently running.
BLUE:	Item has completed running.

2. While an item is running, its Set button will be colored Green, and the Elapsed Time box indicates the how long the item has been running. When the item has finished running, its color will be set to Blue to indicate it has completed running.
3. To turn off the Schedule Mode, click the *Stop* button.

Viewing a Mode while it is running from the schedule

Normally, when an item in the schedule is running, its status is colored Green, and the Elapsed Time indicates how long the item has been running. If desired, you can press the *View Mode* button to display the running item in its native Run Mode window. This will display details about the item that is running, just as if you had manually run the item.

Alternately, checking the *AutoView* checkbox will cause the currently running item to be viewed in its native Run Mode window, just as if the *View Mode* button was pressed. The Run Mode window will automatically pop up as each item in the schedule is run.

Generating Reports

The *Report* button allows the user to print or export the contents of the schedule mode table.
See Section 7 - Generating Reports for further details. The *Report* button is located on the *Close* tab. Click the *Close* tab at the bottom of the screen to access this button.

Saving Changes

When finished running Schedule Mode, click the *Close* button to return to the main menu window (click the Close tab first if the *Close* button is not visible). If any changes have been made, the software will prompt you to save. Click *Yes* to save any changes and return to the main menu.

7. Generating Reports

The software has the ability generate a wide variety of reports. The Report menu contains a list of available reports, such as Gas Library, Cylinder Library, and various Run Mode reports. Reports can be displayed on the screen or sent to a printer. The list of available reports is described below.

Types of Reports Available

Gas Library	Lists all gases in the gas library
Cylinder Library	Shows cylinder name and details including component gases and concentrations. Report can be limited to a specific cylinder, or can include all cylinders.
Cylinder Audit	Shows cylinder name, creation and expiration dates, and number of days left before it expires. Report is sorted in order of expiring cylinders.
Physical Configuration	Shows the details of the instruments internal Mass Flow Controller/Port/Solenoid assignment table, which defines the internal plumbing configuration of the instrument. The report can be for a single instrument or for all instruments.
Instrument Calibration	Shows the mass flow controller calibration tables for the instrument. The report can be for a single instrument or for all instruments.
Port Configuration	Shows the cylinder/port assignments, including port #, cylinder name and gas of interest and its concentration. The report can show the port configuration for a single desktop, or can include all port configurations.
Concentration Mode	Shows the details of a stored concentration mode file, including cylinders, gas concentrations and total flow rate. The report can show a specific concentration mode for the current desktop, all concentration modes for the current desktop, or all concentration modes for all desktops.
Flow Mode	Shows the details of a stored flow mode file, including cylinders and gas flow rates. The report can show a specific flow mode for the current desktop, all flow modes for the current desktop, or all flow modes for all desktops.
Divider Mode	Shows the details of a stored divider mode file, including cylinders, gas concentrations and flowrates, and run times. The report can show a specific divider mode for the current desktop, all divider modes for the current desktop, or all divider modes for all desktops.
Programs	Shows the details of a stored program mode file, including program name and details of each program step. The report can show a specific program mode for the current desktop, all program modes for the current desktop, or all program modes for all desktops.
Schedules	Shows the details of a stored schedule mode file, including schedule name and details of each schedule item and its start date/time. The report can show a specific schedule mode for the current desktop, all schedule modes for the current desktop, or all schedule modes for all desktops.

Report options

There are several reporting options available which specifies which items are to be included in the reports. The "Create Report For" option lets you restrict a report to a specific item, to all items in the current desktop, or to include all items in all desktops.

In order to use the two "Current Desktop" options, you must have a desktop open, otherwise the only option is to generate a report for all desktops. Some reports are not desktop specific, so the "Current Desktop" option will not be available for them.

- To limit the report to a specific item, click the "Current desktop, selection only" option, and select the desired item in the "Selection" box.
- To limit the report to all items in the current desktop, click the "Current desktop, all items" option. The item shown in the "Selection" box is ignored.
- To report on all items in all desktops, click the "All items" option. The item shown in the "Selection" box is ignored.

To view/print a Report:

1. From the Main Menu, select *Report* and then the type of report you want (or click the Report button directly from the Configure/Run Mode you are currently using)
2. Click the Select Report dropdown box to select the type of report you want.
3. Select the desired "Create Report For" and "Selection" options desired.
4. Click *Print* to view the report on the screen.
5. Click the arrow buttons to move between pages. Click the Magnifying Glass Icon to change the view size.
6. Click the Printer icon to send the report to the printer.
7. When finished, click the Close button to return to the Reports menu.
8. Repeat from step 2 to generate additional reports.
9. When done, click Close to return to the Main Menu.

8. Appendix

8.1 Example 1 - Blending Gases

Creating a 75 ppm CO/250 ppm NO blend in a balance of nitrogen

The first step in creating a gas blend is to create a Desktop file. All of the information that you enter regarding the Gas Blend Program will be saved in this file for future use. To create a Desktop:

1. Double click on the Environics icon to open the software. The main menu will be displayed.
2. Click on *File* and then *New* to add the new Desktop file. The software will prompt you for a name for the Desktop. Type **Demo Blend 1**. The instrument type listed is the Series 4000. Since this is correct, click *OK* to save this new desktop file.
3. After you click *OK*, the program returns you to the main menu. The Demo Blend 1 file is open on the desktop.

Before you can begin creating a blending program, you need to make sure that the gases of interest, the CO and NO, are in the Gas Library. The reason for the library is to store characteristics of the various gases. One of the most important characteristics is the K factor. The K factor is the unique characteristic of heat and specific density of an individual gas as compared to air. The K-factor allows the Environics software to differentiate the different gases and to measure the Thermal Mass Flow. To see if carbon monoxide (CO) and nitric oxide (NO) are entered in the library:

1. Click on *Configure* and then *Gas* from the main menu. The Gas Library will be opened.
2. The gases are listed alphabetically. Use the scroll bar on the right hand side of the screen to search down the library for CO and then NO. Both are in the library.

Since we plan to use these two gases as well as nitrogen to create our blend, it will make our work easier if we give these three gases preference. Preferred status takes a gas out of alphabetical order and places it at the top of the list. Once selected, we won't have to scroll down through the library to the gases because they will be the first ones listed. To give the gases preference:

Scroll down to the CO listing and click on the row. The row will turn light blue.
Click on the *Set Preference* button. The gas will be moved to the top of the list for easy selection later on.
Repeat this procedure for NO and nitrogen. Scroll to the top of the list to see that they are now located there.
Once the three gases have been set as preferred gases, click the *Close* button to return to the main menu.

Now that we know the gases are in the library, it is time to set up the Cylinder Library. This is where we tell the computer the contents of each of the cylinders we plan to use. Once entered, the cylinder can be accessed from any desktop file, not just the Demo Blend 1 file we've made. To add our three cylinders in the library:

1. Click on *Configure* and then *Cylinder*. This will open the Cylinder Library menu.
2. First we'll add the nitrogen cylinder to the library. Click the *Add* button at the top of the screen. The software will then prompt for a name for the cylinder. Type **Nitrogen balance**, and click *OK*.
3. The next step is to enter the name of the gas in the Contents table. Using the pull down box under the Name heading of the contents box, click on *Nitrogen* to select this gas. Note that the CO, NO, and nitrogen are listed first. This is because we gave the preference. Once nitrogen is selected, the rest of the table is filled in automatically. Since this is the balance cylinder, it is 100% nitrogen. Both the balance and interest boxes will be checked.
4. Next, you can add the CO cylinder. Click *Add*. The program will ask if you want to save the changes made. Click *Yes* to save the Nitrogen Balance configuration.
5. Proceed by typing **CO 25,000 ppm** at the prompt. Click *OK* to enter this name. The first gas you should enter is always the balance. So using the pull down box under the Name heading of the contents box, click on *Nitrogen* to select this gas. Next, use the pull down box in the next row to select *CO* by clicking on this gas. Since CO is the gas we are interested in, click on the Interest box. A small "x" should appear. Finally, enter the concentration of the CO by clicking on the Concentration box and typing in **25,000**. The units are ppm by default.

6. Finally, we can enter the NO cylinder. This is entered just as the CO cylinder. Click *Add*. Click *Yes* to save the CO 25,000 ppm cylinder.
7. Proceed by typing **NO 10,000 ppm** at the prompt. Click *OK* to enter this name. The first gas you should enter is always the balance. So using the pull down box under the Name heading of the contents box, click on *Nitrogen* to select this gas. Next, use the pull down box in the next row to select *NO* by clicking on this gas. Since NO is the gas we are interested in, click on the Interest box. A small "x" should appear. Finally, enter the concentration of the NO by clicking on the Concentration box and typing in **10,000**. The units are ppm by default.

Notice that after the concentration of the NO or CO is added, the concentration of the nitrogen is automatically calculated. Exit the Cylinder Library menu, click *Close*. The software will prompt you to save the NO cylinder. Click *Yes* to save. This returns you to the main menu.

The final step in preparing to create a Gas Blend is to configure the ports. This tells the computer which cylinder is attached to which port. To configure the ports:

1. Click on *Configure* and then *Port*. This will open the Port Configuration menu. Since each desktop can only have one port configuration, you do not need to create a name for the port configuration as you did with the cylinders. The port configuration is saved as a part of the Demo Blend 1 desktop file.
2. In this example, the nitrogen cylinder is attached to port 1. Using the pull down box under Cylinder in the port 1 row, select the *Nitrogen balance* cylinder. Notice the rest of the columns are automatically filled in based on the information you entered in the Cylinder Library.
3. The NO cylinder is attached to port 3. Using the pull down box under Cylinder in the port 3 row, select the *NO 10,000 ppm* cylinder.
4. Finally, the CO cylinder is attached to port 4. Using the pull down box under Cylinder in the port 4 row, select the *CO 25,000 ppm* cylinder.
5. Once the three cylinders are assigned to the proper ports, click on *Close* to exit the Port Configuration menu. The software will prompt you to save the changes to the port configuration. Click *Yes* to save the changes.

You are now ready to enter the program that will be run to create the 75 ppm CO/ 250 ppm NO blend. Blending is done using the Concentration Mode. To program the Concentration Mode:

1. Click *Run* and *Concentration*. This will open the Concentration Mode menu.
2. The first thing we must do is give the new mode a name. Click *Add*. Type **CO 75ppm, NO 250ppm**. Click *OK* to enter this new mode name.
3. The first cylinder added is always the balance cylinder. Using the pull down box under Cylinder in the Contents table, select the *Nitrogen Balance* cylinder by clicking on it. The balance box is automatically checked. In the next row, select the *CO 25,000 ppm* cylinder using the pull down box under Cylinder. In the third row, select the *NO 10,000 ppm* cylinder using the pull down box under Cylinder.
4. Now that all three cylinders are entered, we have to enter our desired output concentrations in the Target OGC column. Click in the Target OGC box in the CO 25,000 ppm row. Type **75**, since this is the desired final concentration. In the NO 10,000 ppm row, type **250** in the Target OGC column. Finally, click in the Total Flow box at the bottom of the chart. In this example, the diluent flow controller is a 10,000 cc controller. So type **10,000** as the Total Flow. The flows you can use may vary depending on the size of your mass flow controllers.

Your program is now complete. All that's left is to run the mode. The mode can be run in two different ways. You can start and stop the mode manually. This is done by clicking *Run* to start the flow of gas and then *Stop* to stop it.

You can also run the mode for a set time period. This is what we will do.

We will run this blend for 5 minutes. In the Run Time box, type **00:05:00**.

Click *Run* to begin the flow. The elapsed time of the run is shown at the bottom of the screen. The mode will automatically stop after the 5 minutes are up. To stop before that, click *Stop*.

Creating a 100 ppm CO/250 ppm NO blend in a balance of nitrogen using the Copy function

If you have another Gas Blend to prepare that is different by only the concentration of CO, for example, it is very easy to create the second program using the Copy feature. To create a blend of 100 ppm CO and 250 ppm NO with a nitrogen balance:

1. With our first program, CO 75ppm, NO 250ppm still open, click on *Copy*. The software will prompt you to save the original mode. Click *Yes* to save. The software will prompt for a name for the new program. Type **CO 100ppm NO 250ppm**. Click *OK* to enter this new mode name.
2. Notice that this is an exact copy of the CO 75ppm, NO 250ppm mode, except with the new CO 100ppm NO 250ppm name. To change the CO Target OGC from 75 to 100, click on the Target OGC box in the CO 25,000 ppm row. Type **100**. The new mode is now complete and can be run in the same manner as the first mode.

Using the blending of 75 ppm CO and 250 ppm NO in a balance of nitrogen as a guide, any gas blend you desire can be run.

8.2 Example 2 - Diluting Gases (4040 only)

Diluting NO from 10,000 ppm to 250 ppm in a balance of nitrogen

Diluting a gas, like blending gases, requires that the gases being used are in the Gas Library. The first step in diluting a gas is to create a Desktop file. All of the information that you enter regarding the Gas Dilution Program will be saved in this file for future use. To create a Desktop:

Double click on the Environics icon to open the software. The main menu will be displayed.
Click on *File* and then *New* to add the new Desktop file. The software will prompt you for a name for the Desktop. Type **Demo Dilute 1**. The instrument type listed is the Series 4000. Since this is correct, click *OK* to save this new desktop file.
After you click *OK*, the program returns you to the main menu. The Demo Dilute 1 file is open on the desktop.

Before you can begin creating a dilution program, you need to make sure that the gas of interest (NO) and the balance gas (nitrogen) are in the Gas Library. The reason for the library is to store characteristics of the various gases. One of the most important characteristics is the K factor. The K factor is the unique characteristic of heat and specific density of an individual gas as compared to air. The K-factor allows the Environics software to differentiate the different gases and to measure the Thermal Mass Flow. To see if nitric oxide (NO) and nitrogen are entered in the library:

1. Click on *Configure* and then *Gas* from the main menu. The Gas Library will be opened.
2. The gases are listed alphabetically. Use the scroll bar on the right hand side of the screen to search down the library for NO and then Nitrogen. Both are in the library.

Since we plan to use these two gases to create our dilution, it will make our work easier if we give these them preference. Preferred status takes a gas out of alphabetical order and places it at the top of the list. Once selected, we won't have to scroll down through the library to the gases because they will be the first ones listed. To give the gases preference:

Scroll down to the NO listing and *Click* on the row. The row will turn light blue.
Click on the *Set Preference* button. The gas will be moved to the top of the list for easy selection later on.
Repeat this procedure for nitrogen. Scroll to the top of the list to see that they are now located there.
Once the two gases have been set as preferred gases, click the *Close* button to return to the main menu.

Now that we know the gases are in the library, it is time to set up the Cylinder Library. This is where we tell the computer the contents of each of the cylinders we plan to use. Once entered, the cylinder can be accessed from any desktop file, not just the Demo Dilute 1 file we've made. To add our cylinders in the library:

Click on *Configure* and then *Cylinder*. This will open the Cylinder Library menu.
First we'll add the nitrogen cylinder to the library. Click on the *Add* button at the top of the screen. The software will then prompt for a name for the cylinder. Type **Nitrogen balance** and click *OK*.
The next step is to enter the name of the gas in the Contents table. Using the pull down box under the Name heading of the contents box, click on *Nitrogen* to select this gas. Note that the NO and nitrogen are listed first. This is because we gave the preference. Once nitrogen is selected, the rest of the table is filled in automatically. Since this is the balance cylinder, it is 100% nitrogen. Both the balance and interest boxes will be checked.
Next we can enter the NO cylinder. Click *Add*. Click *Yes* to save the Nitrogen Balance cylinder.
Proceed by typing **NO 10,000 ppm** at the prompt. Click *OK* to enter this name. The first gas you should enter is always the balance. So using the pull down box under the Name heading of the contents box, click on *Nitrogen* to select this gas. Next, use the pull down box in the next row to select *NO* by clicking on this gas. Since NO is the gas we are interested in, click on the Interest box. A small "x" should appear. Finally, enter the concentration of the NO by clicking on the Concentration box and typing in **10,000**. The units are ppm by default.

Notice that after the concentration of the NO is added, the concentration of the nitrogen is automatically calculated. Exit the Cylinder Library menu, click *Close*. The software will prompt you to save the NO cylinder. Click *Yes* to save. This returns you to the main menu.

The final step in preparing to create a Dilution Program is to configure the ports. This tells the computer which cylinder is attached to which port. To configure the ports:

1. Click on *Configure* and then *Port*. This will open the Port Configuration menu. Since each desktop can only have one port configuration, you do not need to create a name for the port configuration as you did with the cylinders. The port configuration is saved as a part of the Demo Dilute 1 desktop file.
2. In this example, the nitrogen cylinder is attached to port 1. Using the pull down box under Cylinder in the port 1 row, select the *Nitrogen balance* cylinder. Notice the rest of the columns are automatically filled in based on the information you entered in the Cylinder Library.
3. The NO cylinder is attached to port 3. Using the pull down box under Cylinder in the port 3 row, select the *NO 10,000 ppm* cylinder.
4. Once the two cylinders are assigned to the proper ports, click on *Close* to exit the Port Configuration menu. The software will prompt you to save the changes to the port configuration. Click *Yes* to save the changes.

You are now ready to enter the program that will be run to create the 250 ppm NO from the NO 10,000 ppm cylinder. Diluting is done using the Concentration Mode. To program the Concentration Mode:

Click *Run* and *Concentration*. This will open the Concentration Mode menu.
The first thing we must do is give the new mode a name. Click *Add*. Type **Dil NO 250 ppm**. Click *OK* to enter this new mode name.
The first cylinder added is always the balance cylinder. Using the pull down box under Cylinder in the Contents table, select the *Nitrogen Balance* cylinder by clicking on it. The balance box is automatically checked. In the next row, select the *NO 10,000 ppm* cylinder using the pull down box under Cylinder.
Now that both cylinders are entered, we have to enter our desired output concentration in the Target OGC column. Click in the Target OGC box in NO 10,000 ppm row. Type **250** in the Target OGC column. Next, enter the total output flow by clicking in the Total Flow box at the bottom of the chart. In this example, the diluent flow controller is a 10,000 cc controller. So type **10,000** as the Total Flow. The flows you can use may vary depending on the size of your mass flow controllers.

Your Program is now complete. All that's left is to run the mode. The mode can be run in two different ways. You can start and stop the mode manually. This is done by clicking *Run* to start the flow and *Stop* to terminate the mode.

You can also run the mode for a set time period. This is what we will do.
1. We will run this dilution for 5 minutes. In the Run Time box, type **00:05:00**. This is 5 minutes.
2. Click on *Run* to begin the flow. The elapsed time of the run is shown at the bottom of the screen. The mode will automatically stop after the 5 minutes are up. To stop before that, click *Stop*.

8.3 Example 3 - Dividing Gases

Dividing Carbon Monoxide from 25,000 ppm to 25 ppm in 10 user selectable steps (1000 - 1 range)

Dividing a gas requires that the gases being used are in the Gas Library. The first step in dividing a gas is to create a Desktop file. All of the information that you enter regarding the Gas Divider Program will be saved in this file for future use. To create a Desktop:

1. Double click on the Environics icon to open the software. The main menu will be displayed.
2. Click on *File* and then *New* to add the new Desktop file. The software will prompt you for a name for the Desktop. Type **Demo Divide 1**. The instrument type listed is the Series 4040. This is correct since only the Series 4040 has a Divider Mode. Using the pull down box, change the instrument choice from 4000 to 4040. Click *OK* to save this new desktop file.
3. After you click *OK*, the program returns you to the main menu. The Demo Divide 1 file is open on the desktop.

Before you can begin creating a Divider Mode, you need to make sure that the gas of interest, the CO, and the balance gas, the nitrogen, are in the Gas Library. The reason for the library is to store characteristics of the various gases. One of the most important characteristics is the K factor. The K factor is the unique characteristic of heat and specific density of an individual gas as compared to air. The K-factor allows the Environics software to differentiate the different gases and to measure the Thermal Mass Flow.

To see if carbon monoxide (CO) and the nitrogen are entered in the library:

Click on *Configure* and then *Gas* from the main menu. The Gas Library will be opened.
The gases are listed alphabetically. Use the scroll bar on the right hand side of the screen to search down the library for CO and then Nitrogen. Both are in the library.

Since we plan to use these two gases to create our dilutions, it will make our work easier if we give these them preference. Preferred status takes a gas out of alphabetical order and places it at the top of the list. Once selected, we won't have to scroll down through the library to the gases because they will be the first ones listed. To give the gases preference:

1. Scroll down to the CO listing and click on the row. The row will turn light blue.
2. Click on the *Set Preference* button. The gas will be moved to the top of the list for easy selection later on.
3. Repeat this procedure for nitrogen. Scroll to the top of the list to see that they are now located there.
4. Once the two gases have been set as preferred gases, click the *Close* button to return to the main menu.

Now that we know the gases are in the library, it is time to set up the Cylinder Library. This is where we tell the computer the contents of each of the cylinders we plan to use. Once entered, the cylinder can be accessed from any desktop file, not just the Demo Divide 1 file we've made. To add our cylinders in the library:

Click on *Configure* and then *Cylinder*. This will open the Cylinder Library menu.
First we'll add the nitrogen cylinder to the library. Click on the *Add* button at the top of the screen. The software will then prompt for a name for the cylinder. Type **Nitrogen balance**. Click *OK* to enter this name.
The next step is to enter the name of the gas in the Contents table. Using the pull down box under the Name heading of the contents box, click on *Nitrogen* to select this gas. Note that the CO and nitrogen are listed first. This is because we gave the preference. Once nitrogen is selected, the rest of the table is filled in automatically. Since this is the balance cylinder, it is 100% nitrogen. Both the balance and interest boxes will be checked.
Next we can enter the CO cylinder. Click *Add*. Click *Yes* to save the Nitrogen Balance cylinder.
Proceed by typing **CO 25,000 ppm** at the prompt. Click *OK* to enter this name. The first gas you should enter is always the balance. So using the pull down box under the Name heading of the contents box, click on *Nitrogen* to select this gas. Next, use the pull down box in the next row to select *CO* by clicking on this gas. Since CO is the gas we are interested in, click on the Interest box. A small "x" should appear. Finally, enter the concentration of the CO by clicking on the Concentration box and typing in **25,000**. The units are ppm by default.

Notice that after the concentration of the CO is added, the concentration of the nitrogen is automatically calculated. Exit the Cylinder Library menu, click *Close*. The software will prompt you to save the CO cylinder. Click *Yes* to save. This returns you to the main menu.

The final step in preparing to create a Divider Program is to configure the ports. This tells the computer which cylinder is attached to which port. To configure the ports:

1. Click on *Configure* and then *Port*. This will open the Port Configuration menu. Since each desktop can only have one port configuration, you do not need to create a name for the port configuration as you did with the cylinders. The port configuration is saved as a part of the Demo Divider 1 desktop file.
2. In this example, the nitrogen cylinder is attached to port 1. Using the pull down box under Cylinder in the port 1 row, select the *Nitrogen balance* cylinder. Notice the rest of the columns are automatically filled in based on the information you entered in the Cylinder Library.
3. The CO cylinder is attached to port 4. Using the pull down box under Cylinder in the port 4 row, select the *CO 25,000 ppm* cylinder.
4. Once the two cylinders are assigned to the proper ports, click on *Close* to exit the Port Configuration menu. The software will prompt you to save the changes to the port configuration. Click *Yes* to save the changes.

After the port assignments are made, you are ready to program the Divider Mode to generate the desired 10 step division of the CO from 25,000 ppm to 25 pp in 10 user defined steps. The Divider Mode establishes 10 concentration steps automatically as a percentage of the cylinder. They are 100%, 90%, 80%, 70%, 60%, 50%, 40%, 30%, 20%, and 10%. These steps can be changed by the user to any percentage. Once you are satisfied with the divide mode steps, you just need to enter the cylinders you want to use, the run time for each step, and the target flow.

Click on *Run* and *Divider*. This will open the Divider Mode menu.

The first thing we must do is give the new mode a name. Click *Add*. Type **Div CO 250 ppm**. Click *OK* to enter this new mode name.

The Nitrogen Balance cylinder, assigned to port 1, is automatically selected as the Dilution gas. This is why we must assign the balance gas to this port. The CO 25,000 ppm cylinder is the only other cylinder assigned to a port, so it is automatically selected as the Span gas.

With both cylinders automatically entered, we can now set up the 10-step division mode. Click on *Init*. The software will prompt for a Run Time. We will run each step for 5 minutes. Click in the Run Time box and type **00:05:00** for 5 minutes. The program also prompts for a total flow. We will run at a total flow of 10,000 cc. Click in the Total Flow box and type **10,000**. Click *OK* to enter these values. The program automatically sets up the ten equal steps for the 10-step dilution.

With the Divider Mode programmed, all that remains is to run the program. The mode can be run in two different ways, automatically or manually.

To Automatically Run the Divider Mode:

1. Click on the *Run tab* to access the Run menu.
2. To start the run, click on the *Auto* button. The instrument will start at step 1 and automatically run through all 10 steps. Each step will take 5 minutes to run. The run time counter shows the total elapsed time.
3. To stop the current step and advance to the next, click the *Advance* button.
4. The instrument will stop automatically when all 10 steps are executed. To stop the instrument before this point, click the *Stop* button.

To Manually Run the Divider Mode:

Click on the *Run tab* to access the Run menu.

Select the desired step by clicking on it.

Set the desired run time for the step by clicking in the Run Time box and typing in **00:05:00**. Click the *Manual* button. The instrument will run for the specified length of time and then stop. While the instrument is running, the run status box indicates the elapsed run time.

To stop the mode before the set run time is complete, click on the *Stop* button.

8.4 MFC Calibration Procedure

The following procedure should be followed for calibration and re-entry of data for a Series 4000 instrument.

MFC Zero Check

Connect the (-) lead from a voltmeter to PC402 GROUND (TP2).
Connect the (+) lead to each MFC Response Test Point:
MFC1 (TP9), MFC2 (TP11), MFC3 (TP13), MFC4 (TP15).
Verify that the voltage at each test point is within +/- .025v from 0 volts for a Tylan MFC and +0.005v to 0.015v for a Hastings MFC. If the "zero" is out of spec, the MFC “zero” pot must be adjusted to bring it within specs.

NOTE: Some MFCs require the cover to be removed to make adjustments. In this case, after performing “zero” adjustment, replace MFC cover. After adjusting “zero” pot, replace system cover and wait at least 1 hour for temperature to stabilize before proceeding to MFC Calibration.

MFC Calibration

1. Connect calibration gas to the appropriate input ports for the MFCs being calibrated. The input pressure should be approximately 25 PSIG.
2. Connect a primary standard flow calibration device to the output of the system.
3. Be sure the instrument is connected to the PC and powered up.
4. Run the Environics Series 4000 software.
5. From the menu screen, select **Configure**
6. Select **Instrument**
7. Select **Calibrate**
8. In the **Reference Gas** box, click the down arrow button and set this to match the calibration gas for that MFC.
9. In the **Controller Instrument** box, click the down arrow button, and select the serial number of the instrument to be calibrated.
10. In the **MFC** box, click the down arrow button, and select the desired Mass Flow Controller to enter data for (start with MFC 1). NOTE: If the MFC has been changed, Select **Init Points**, enter appropriate size in **Max Flow**, Table will initialize.
11. Click on the 100% calibration point and press **Start**. Wait at least 10 minutes for flow to stabilize before continuing.
12. Take a flow measurement. Measured flow value must be between 100% and 110% of the MFCs rated full-scale flow. If flow value is out of spec, MFC “span” pot must be adjusted.

    NOTE: Some MFCs require the cover to be removed to make adjustments. In this case, after performing “span” adjustment, replace MFC cover. After adjusting “span” pot, replace system cover and wait at least 1 hour for temperature to stabilize before continuing.

If measured flow value is acceptable, enter the measured value in the **True Flow** column. If not, repeat step 11.
Click on the next calibration point, and press the **Start** button. Wait at least 2 minutes for flow to stabilize before continuing.
Take a flow measurement, and enter the measurement value in the **True Flow** column.
Repeat steps 14-15 until all calibration points have been completed.
Press **Stop**, then press **Save** to save the MFC calibration data. This will down load the new calibration files to the instrument.
NOTE: If a warning appears “CAL TABLE ERROR -3” click OK, then save again before exiting
Repeat steps 10-17 until all MFCs have been calibrated
Click **Close** to return to the main menu screen

Entry of the Calibration Data is now complete.

8.5 Working with multiple calibration tables

If you have ordered your systems Mass Flow Controllers to be calibrated in more than one gas, this section will show the steps needed to switch between calibration tables. Be sure you have installed the additional Instrument Data Disks provided (1 per flow controller calibration) which contain the additional MFC flow calibration data. Refer to *Section 2.3 - Instrument Data Installation* for more information.

To switch one or more Flow Controller calibration tables:

From the Main menu, select *Configure – Instrument - Calibrate*
In the *Controller Instrument* box, select the serial number for the instrument you are using.
In the *MFC* box, click the down arrow button, and select the desired Mass Flow Controller whose calibration table is to be changed.
In the *Reference Gas* box, click the down arrow button and set this to match the calibration gas for that MFC.
In the *Table Selection* section at the bottom of the screen, click *Select*.
This will bring up the Select Calibration Table window. Click in the *Description* field and select the desired calibration table. The description should include the name of the calibration gas used to calibrate the flow controller. Click *OK*.
The *Reference Gas* will now show the same calibration gas as listed in the *Description* field.
In the *Table Selection* section at the bottom of the screen, click *Save*. This will update the instrument's internally stored calibration data.
Repeat from steps 3 for each Mass Flow Controller whose calibration table you want to change.
Click *Close* to return to the main menu screen.

NOTES