MMS 405

Operator's Handbook

Issue V1.0

The Business of Science®

OXFORD INSTRUMENTS

Copyright

Disclaimer

Oxford Instruments' policy is one of continued improvement. The Company reserves the right to alter without notice the specification, design or conditions of supply of any of its products or services. Although every effort has been made to ensure that the information in this manual is accurate and up to date, errors may occur. Oxford Instruments NanoScience shall have no liability arising from the use of or reliance by any party on the contents of this manual and, to the fullest extent permitted by law, excludes all liability for loss or damages howsoever caused.

Trademarks

The Oxford Instruments Logo, Oxford Instruments Direct and Cryofree are trademarks or registered trademarks of Oxford Instruments plc or its subsidiaries. The use of our trademarks is strictly controlled and monitored and any unauthorised use is forbidden.

Other trademarks and registered trademarks not listed above may be used in this manual.

Technical and Customer Support

Contact details for Technical and Customer Support can be found on page 43.

Oxford Instruments NanoScience
Tubney Wood, Abingdon, Oxon, OX13 5QX, England
Tel: +44 (0)1865 393 200
Fax: +44 (0)1865 393 333
E-mail: nanoscience@oxinst.co.uk
www.oxford-instruments.com

Contents

Safety Issues 6
- Safety symbols used in this manual 6
- Disclaimer 6
- Warnings 6

Introduction 7
- MMS Features 7
- Requirements 7
- Scope of this document 8
- Comparison with Objectbench 8

Installing MMS 9
- The Software Installation Program 9

Running MMS 10
- MMS workspace 10
- Getting Help 11
- MMS menu bar 12
  - System menu 12
    - Menu Item: Connect All 12
    - Menu Item: Disconnect All 13
    - Menu Item: Exit… 13
  - Windows menu 13
    - Menu Item: XY plot 14
    - Menu Item: Graphical Analysis 17
    - Menu Item: Basic window 25
    - Menu Item: Edit file 25
    - Menu Item: Measurement Sequencer 25
  - Configure menu 26
    - Menu Item: Add/remove instruments 27
    - Menu Item: Signal links 28
    - Menu Item: Instrument links 29
    - Menu Item: Directories 29
    - Menu Item: Graph colours and fonts 29
    - Menu Item: Window font 29
    - Menu Item: Customise menus 29
    - Menu Item: Customise buttons 31
    - Menu Item: View button bar 32
  - Help menu 32
- MMS Button bar 33

Tutorials 34
- Log Temperature controller using XY plot 34
- Set up and control a small group of ISOBUS instruments 35
- Using third party instruments 36

Customer Support 43
Warranty 43
Technical support..43
Europe, RoW, Main Office 43
USA Office..43
Japan Office..43
China Office 43
Additional services available from Oxford Instruments..43
About Oxford Instruments 43
Oxford Instruments NanoScience Limited 44

Appendix 1: Data file format..45

Appendix 2: Instrument Drivers..46
Interfacing Instruments to MMS..46
ITC503 Temperature Controller 48
Preparing to Use the ITC503 from MMS..48
Operating the ITC503 from MMS 49
The ITC503 General Setup Dialog..50
The ITC503 RAM Setup Dialog..51
The ITC503 Range Design Dialog..52
IPS120-10 Magnet Power Supply 54
Preparing to Use the IPS120-10 from MMS..54
Operating the IPS120-10 from MMS..55
The IPS120-10 Setup Dialog 56
ILM Intelligent Level Meter..56
Preparing to Use the ILM from MMS..57
Operating the ILM from MMS 57
The ILM Setup Dialog..58
Heliox Virtual Instrument..58
Connecting the Heliox Virtual Instrument..58
Using the Heliox Virtual Instrument..60
Heliox Setup dialog 62
^{3}He Recondensation..66

Appendix 3: Oxford Instruments ISOBUS..69

Appendix 4: Troubleshooting..71

Welcome

Thank you for choosing your equipment from Oxford Instruments, a company dedicated to providing world-class products and customer support. Our highly trained teams are available to help you with all your queries relating to your order, delivery or technical issues.

As an Oxford Instruments customer, you have access to a worldwide service and support package providing telephone and on-site technical and repair services. In the unlikely event that your product should require repair, our technicians will initiate service under the terms of your Oxford Instruments warranty.

At Oxford Instruments we know that your expectations are at the highest level. We aim to meet and exceed those expectations in the service that we provide, and in the quality you will see when you use Oxford Instruments equipment.

We are delighted you selected Oxford Instruments as your supplier and wish you success with your new equipment.

Jim Hutchins, Managing Director, Oxford Instruments NanoScience

Safety Issues

Before you attempt to install or operate MagLab Measurement Software (MMS), please make sure that you are aware of all safety precautions listed in this document together with the warnings and cautions set out in the separate instrument manuals. If the software is supplied with a superconducting magnet system, you should also read the safety precautions listed in the manuals supplied for the magnet system components.

Safety procedures are vital to prevent

- Serious injury or death
- Serious damage to the equipment.

Cryogenic systems are potentially hazardous and you must take precautions to ensure your own safety, particularly if you are writing your own system control routines.

The general safety precautions required when working with cryogenic systems and superconducting magnets are given in the *Safety Matters* booklet that will have accompanied your Oxford cryogenic system.

Safety symbols used in this manual

Symbols are used in this manual to draw your attention to safety procedures that you must follow to protect yourself or the equipment. There are two types of hazard symbol used in this manual:

Warning: The warning triangle highlights dangers which may cause injury or, in extreme circumstances, death. Warnings and cautions must be followed to ensure your own safety.

Caution: The general caution symbol highlights actions that you must take to prevent damage to the equipment. The action is explained in the text.

Disclaimer

Oxford Instruments cannot accept responsibility for damage to the system caused by failure to observe the correct procedures laid down in this manual and the other manuals supplied with the system. The warranty may be affected if the system is misused, or the recommendations in the manuals are not followed.

Warnings

Warnings and cautions must be followed to ensure your own safety.

Introduction

Oxford Instruments MagLab Measurement Software (MMS) is a general purpose program for controlling all instrumentation from a PC. Possible instrumentation includes temperature controllers, Cryogen level meters, and magnet power supplies manufactured by Oxford Instruments. It makes use of the features of Microsoft Windows™ to provide a flexible yet simple user interface. It uses standard hardware interfaces, such as RS232, *Oxford Instruments* ISOBUS or GPIB, to control instrumentation remotely. The Measurement Engineer can be used to create sequences to perform a series of operations and data acquisitions. Data acquired in this way may be displayed graphically, printed, and stored in data files.

Note: Users are responsible for ensuring that MMS operates normally and should take reasonable care that hardware will not be subject to demands outside working specifications.

This manual does not contain safety information for the instruments or systems that this software can be used to control. For this information, please refer to individual instrument manuals and system documentation.

MMS Features

- Here is a brief summary of the MagLab Measurement Software features:

- Remote control of electronic instruments supplied by Oxford Instruments using a software front panel that simulates the real front panel.
- Measurement Sequencer for creating sequences of measurements without programming.
- A wizard for creating measurement engineer actions for communicating with third party instruments.
- The ability to monitor signals derived from the instruments in a very flexible way. Data may be displayed graphically and stored in data files.
- Powerful built-in BASIC interpreter. A source code debugger makes development easy, while a dialog editor allows custom dialogs boxes to be created.
- Data may be stored in ASCII text files, and retrieved for analysis or printing.
- Complete named system configurations may be saved so that a number of different experiments may be stored and retrieved separately.

Requirements

- Ideally MMS requires a PC with Microsoft Windows98 installed. However, it can be installed on Windows XP and run in Win98 compatibility mode.

MMS is only supported for Windows 98. It will generally also work with Windows XP. MMS's internal GPIB driver cannot be used with Windows XP.

Scope of this document

Use this manual to

- Provide you with an introduction to MMS philosophy and features
- Help you install MMS on your PC

Use on-line Help for immediate access to reference data for all the instrument drivers.

Use the Sequencer Manual for advice on writing (measurement) sequences

Comparison with Objectbench

Some of the key improvements are:

- Measurement Sequencer has been introduced, providing the means to design experiments without programming experience (replaces macro engineer)
- Menus and buttons may now be customised
- For advanced users the Basic debugger has been greatly extended

In order to reflect these major enhancements the software package has been renamed MagLab Measurement Software (MMS).

Installing MMS

The Software Installation Program

The installation process is similar in both cases. Various choices are presented during the installation process and this section gives guidance. Instructions for loading MMS from the CD-ROM are also held in a file readme.txt on the CD. Read this first.

Insert the CD. If the CD does not autorun use Explorer to navigate to the CD folder and run “setup”.

When the installation program is run, the dialog shown in Figure 1 is displayed.

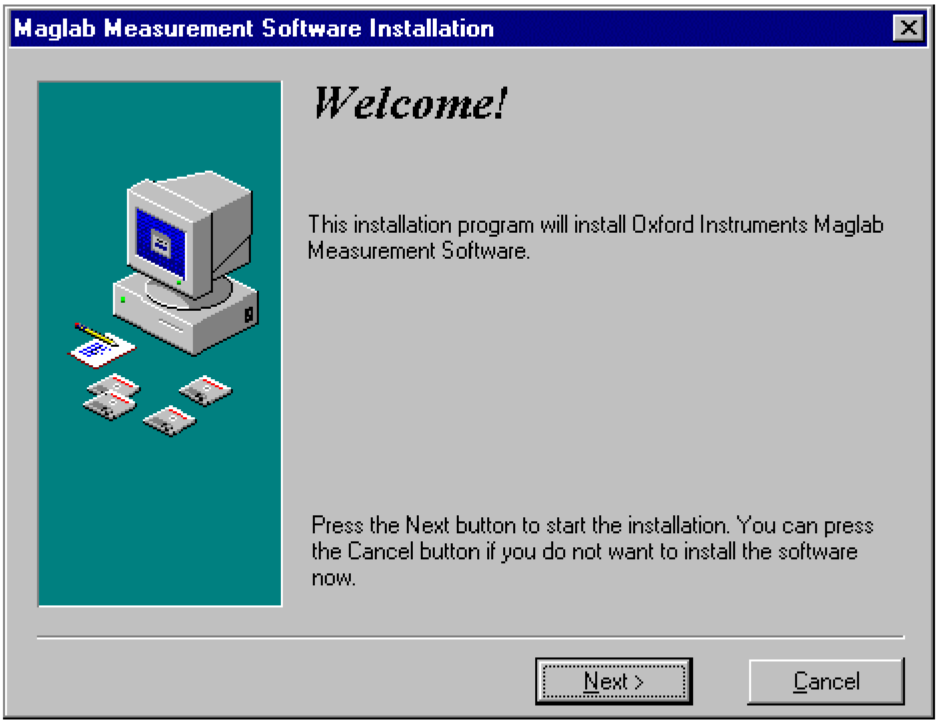


Figure 1 The first installation dialog.

From this point, follow the instructions given by the installer. The default program file directory is C:\Program Files\MMS. The default data file directory is C:\MMSdata.

Running MMS

To run MagLab Measurement Software, click the Windows Start button and find the Oxford Instruments item within the Programs menu. Select MMS. Alternatively, use Windows to create a short cut to *ob32.exe*.

MMS workspace

Running MMS for the first time the main window display is similar to that shown in Figure 2.

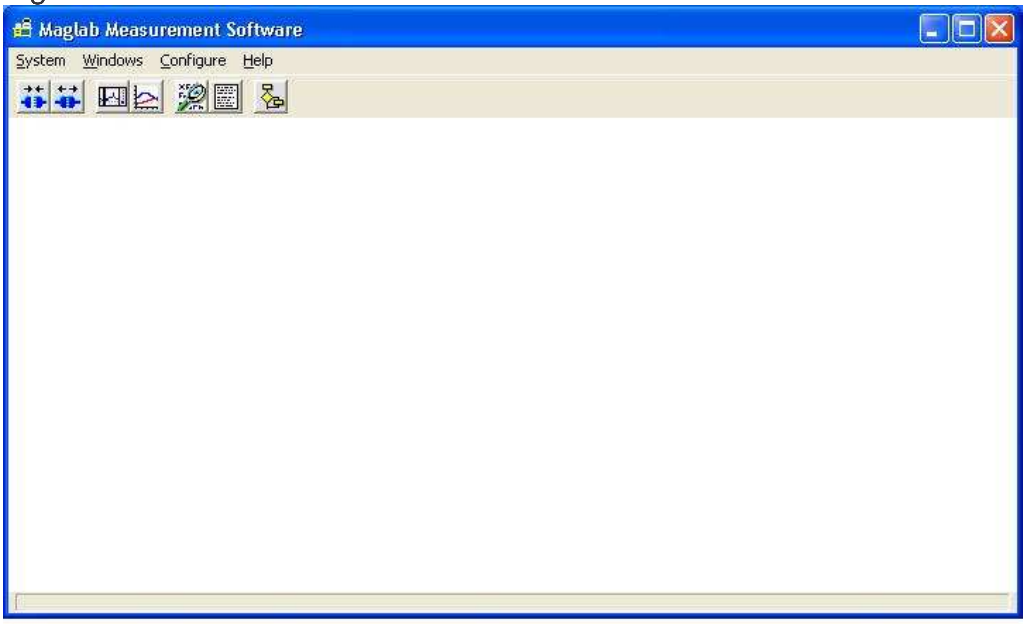


Figure 2 MMS Main window

The Main window consists of a menu bar and button bar. Most MMS functions can be accessed using the menus. Some frequently used menu items are duplicated in the button bar with the row of 7 icons (Figure 3).

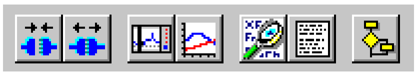


Figure 3 Buttons that duplicate menu items

The main part of the workspace will display features such as graphs, instrument drivers and dialogs, as required by the user. Figure 4 gives a typical example.

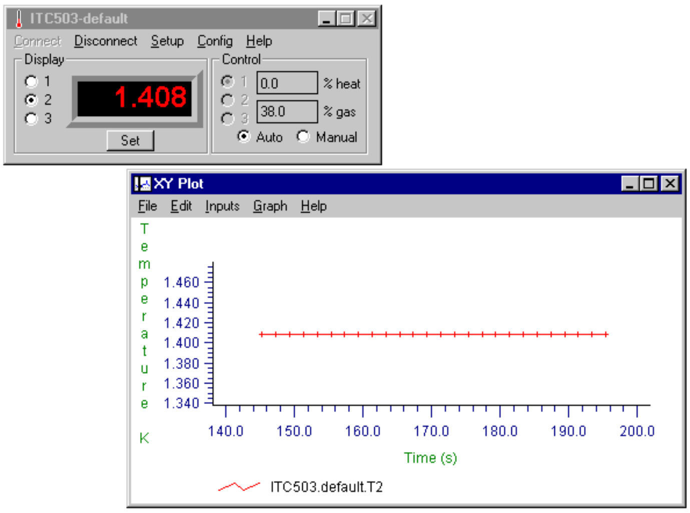


Figure 4 A typical MMS workspace

Most windows, including those for instrument drivers and graphs, may be minimised and restored using the standard Windows icons at the top right of the window. Various functions are not permitted and are greyed out; for instance, you may not resize or delete an instrument driver.

Figure 5 MMS minimised windows

Getting Help

Use the main HELP menu to access on-line Help for MMS, either through a top level Contents page or by searching for a keyword. Many windows will also have individual Help menus which access the same Contents page or specific Help for that window. Help buttons appear throughout MMS for context sensitive Help.

MMS menu bar

System menu

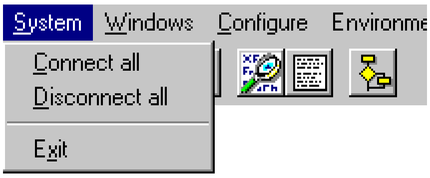


Figure 6 System menu

Menu Item: Connect All

This connects all the instrument drivers to the real (physical) instruments. Each instrument driver becomes "live". The instrument driver may then be controlled as if it was the real instrument. Further details for each instrument are given in Appendix 2: Instrument Drivers.

You need to be familiar with the operation of each instrument (refer to individual Operator's manuals) as MMS does not provide any special safety features.

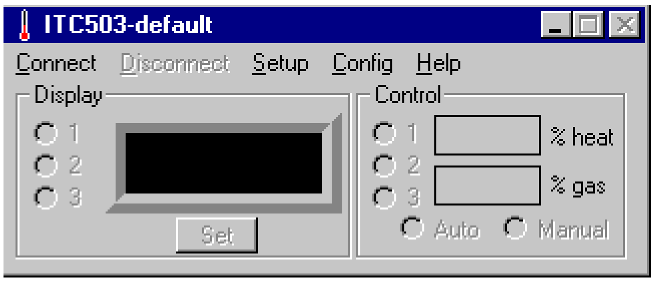


Figure 7 Disconnected instrument driver

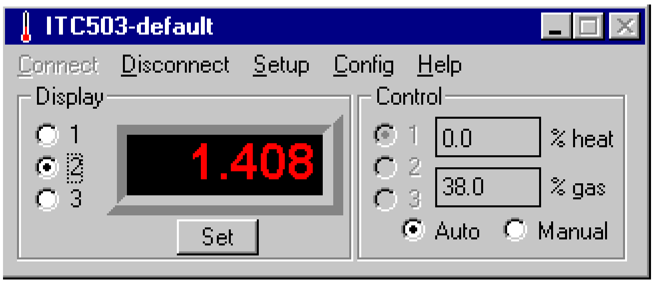


Figure 8 Connected instrument driver

If you receive the following error message.

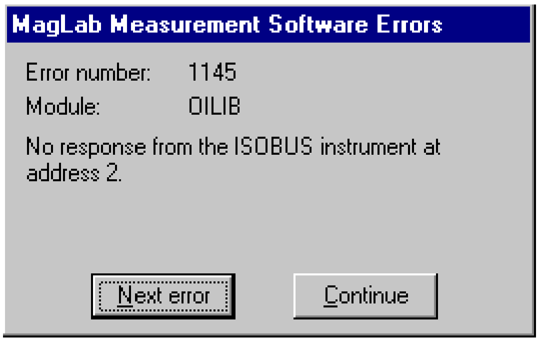


Figure 9 Connect failure dialog

Then the instrument is switched off, not electrically connected to the PC or the connection is not correctly configured (see Appendix 4: Troubleshooting). Investigate and try again.

Menu Item: Disconnect All

This disconnects all the instrument drivers from the real (physical) instruments. This will happen automatically if you close or exit MMS.

Menu Item: Exit….

Use this command to close down the software. All open files are closed and all instruments are disconnected. SetUp options are saved so that the next time MMS is run the same windows with the same settings are present. Customised menus and buttons are also saved.

Closing/Exiting MMS is the recommended way to save *any* ***customised settings. You should use it regularly if you are configuring MMS.***

Windows menu

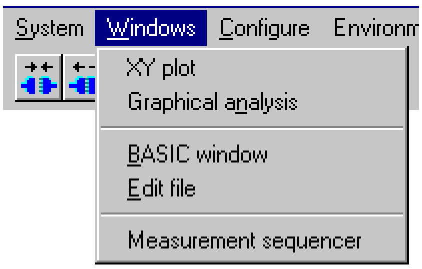


Figure 10 Function windows menu

Menu Item: XY plot

When one or more instruments have been loaded as described in previous sections, it is possible to plot graphs of signals available from the instruments (for example, temperature or field strength). Before attempting this, refer to later sections of this manual for information on how to control your particular instruments, and "connect" them so that they are obtaining measurements in real time.

Any number of different XY plot windows may be opened and in operation at any time. However, there is an upper limit of 1000 data points allowed in any one graph. After this limit has been exceeded, early data points will be discarded as new data are added. This is done to keep the display speed at an acceptable level. If it is necessary to record more than 1000 data points use a log file to store all the data as described in section 0. This file may subsequently be loaded into an analysis window, which is capable of displaying any number of points.

To create a new XY plot window, press  or select the "XY Plot" option from the "Windows" main menu option. You may close existing windows by double clicking the top left corner of the window.

The appearance of the graph can be changed at any time as described in Menu Item: Graphical Analysis.

Selecting the Signals to Plot

The XY plot window is a window containing a graph. It may be moved or resized to any position on the screen. Initially an empty graph is displayed, ready to receive data as shown in Figure 11.

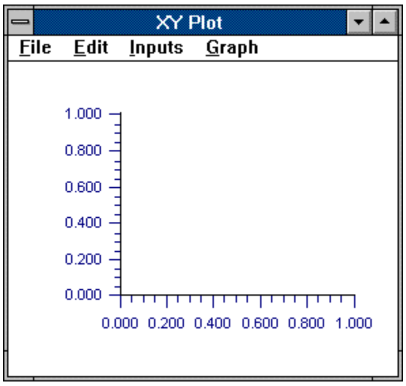


Figure 11 An empty XY Plot window.

The most important menu option is "Inputs", because this is used to select which signals are to be plotted, and when. There are three decisions to make: the choice of x parameter; the choice(s) of y parameter(s); and the trigger.

The trigger is the event that causes a point to be plotted on the graph. There are two options. Data can be read and plotted at regular time intervals or it can be plotted every time an instrument driver makes a new signal value available.

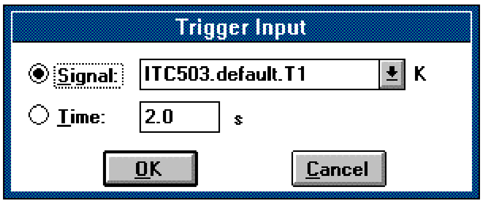


Figure 12 Selecting the trigger for an XY plot.

Figure 12 shows the dialog which is displayed when the Inputs/Trigger menu option is selected. The choice of signal or time trigger may be made by clicking one of the radio buttons on the left of the dialog box. If signal triggering is selected, a signal must be selected from the drop-down list. In this example, signal itc503.default.T1 has been selected. Click on "OK" to confirm this choice.

Having chosen the trigger source, you may now select the signal to be used as the x axis. Select the "Inputs"/"x signal" menu option.

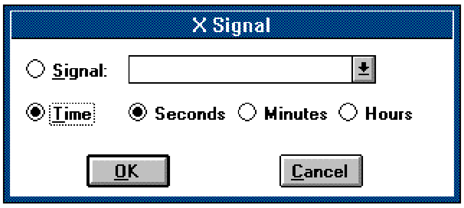


Figure 13 Choice of the signal to display on the x axis of a plot.

Figure 13 shows the dialog box used to select the x parameter. There is a choice between using elapsed time or an instrument signal. As before, click on a radio button to make your choice, and select a signal from the drop-down list if necessary. In this example, time has been chosen for plotting on the x axis, with units of seconds.

Finally, choose the signals to be plotted on the y axis. Select the Inputs/y signals option to display the dialog box shown in Figure 14.

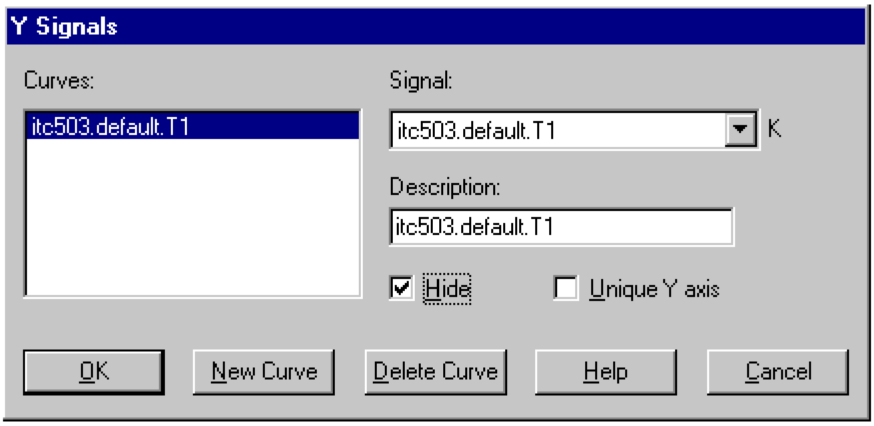


Figure 14 The dialog box used to select y signals in an XY plot.

This dialog allows one or more signals to be selected for use as y co-ordinates of plotted points. To create a new curve (that is a plot of y vs. x) click the "New Curve" button with the mouse. This will cause a new entry to appear in the list of curves with the text "new curve". Select this curve by clicking on it with the mouse, and then use the drop-down list of signals (to the right) to select which signal should be used. When a signal has been selected, the entry in the list of curves will change to a description of the curve, in this example, "itc503.default.T1". If you wish, you may change the description by editing it in the edit box labelled "Description" (to the right). This description will appear in the graph key. When you have created all the curves you require, click on the "OK" button.

There are two check boxes, which provide access to two further features in this dialog box. Selecting "Hide" will make the selected curve invisible, without deleting the data. It may later be displayed by clicking on "Hide" again. This feature can be useful if you wish to omit one or more curves when the graph is printed.

Selecting "Unique y axis" forces the graph to include a separate y axis for this curve. Normally, the software attempts to share y axis between curves if the points plotted have the same units, creating one y axis for each set of units (for example field and temperature). When "Unique y axis" is selected, this feature is overridden.

"Connecting" the XY Plot

Once the signals to be plotted have been selected as described in the previous section, select the "Signals"/"Connect" menu option to start plotting data. The relevant instruments must also be connected before data can be acquired. You can press  to connect all the graphs and all the instruments, or use the "System"/"Connect all" menu option.

While the plot is connected, it is no longer possible to change the signals chosen for plotting.

To disconnect the XY plot, select the "Signals"/"Connect" option again, or the main menu option "System"/"Disconnect all". You can disconnect all XY plots and all instruments by pressing .
Sometimes it is useful to clear all old data from the graph before acquiring new data. This may be accomplished using the "Graph"/"Clear data" menu option.

Saving data to a file

You may store the data points recorded in an XY graph in a data file. There are two ways to save the data.

The first is a simple save of the graph displayed, which is performed by using the XY graph's "File|Save" menu option. You are prompted for a file name and the data is saved in the file. The data may be viewed at a later date by opening an analysis window and loading the data file. Note that this method of saving data saves exactly the data visible in the graph, so if the limit of 1000 points plotted in the graph has been exceeded, and old data lost, the old data is not saved to the file either. To be sure of not losing old data, you should instead save data to a file using the "Log" option.

When logging data to file, each data point is written to the file as it is read from hardware and plotted on the graph. File data are never discarded. Be aware that it is possible to create very large files in this way. To log data to a file, use the "File|Log" menu options. You will be prompted for a file name, and then logging will commence. To check that data is being logging, look at the "File" menu. You should see a small tick beside the "Log" option. To stop logging, select this option again. The tick will disappear.

Menu Item: Graphical Analysis

MMS displays data in graphical format in XY plots, in analysis windows and in graph windows generated by the Measurement sequencer. In each case a set of operations may be performed on the graph, such as rescaling. These operations are described in this section of the manual.

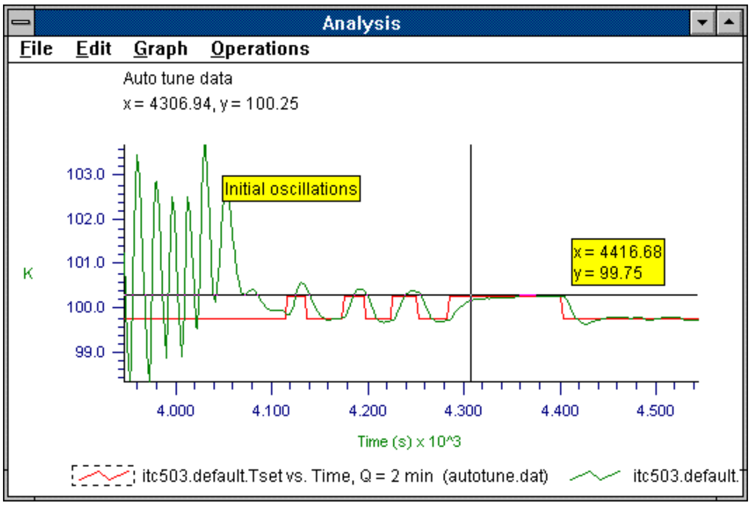


Figure 15 A Typical Graph.

A typical graph is shown in Figure 15. A key appears beneath the graph, which contains an entry for every curve in the graph; the key for the currently selected curve is surrounded by a dashed rectangle as shown. There is a cross (the readout cursor) on the graph which applies to the currently selected curve. The co-ordinates of the cross hair appear above the graph. Above the cross hair co-ordinates there is some descriptive text, in this case “Auto tune data”.

The graph can be manipulated using the mouse, the main menu, and a popup menu accessed through the right mouse button. All these features are described in the following sections.

Mouse Operations

As the mouse pointer is moved around over the graph window, it changes from the usual Windows mouse pointer when it has a special function. Several features require you to drag the mouse. The mouse is dragged by pressing the left mouse button at the starting position, moving the mouse to the finishing position, and releasing the button. The mouse is double clicked by rapidly pressing and releasing the left mouse button twice. A popup menu, the graph popup menu, is accessed using the mouse as described in the table. Popup menus are used most easily by pressing the right mouse button to display the menu, moving the mouse to the menu option you require while still holding the button down, and finally releasing the button to select the option. To select no option, release the right button when the mouse pointer is outside the menu. The menu options are described in subsequent sections of this manual.

Here are the different possibilities:

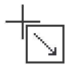
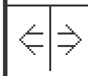
	The mouse pointer is over a graph and can be used to rescale the graph by zooming in on a selected rectangular region. Drag the mouse to select a rectangular outline on the graph. Clicking with the left mouse button places the readout cursor at the position on its curve nearest to the position where the mouse was clicked. Clicking with the right mouse button displays the graph popup menu (see section 0).
	The mouse pointer is over the key for a particular curve. Clicking with left mouse button on a curve's key selects that curve. If the curve is already selected, all curves are deselected. Clicking with the right mouse button displays the curve popup menu (see section 0). Double clicking on the descriptive text allows the text to be edited.
	The mouse pointer is over the readout cursor, and can be used to drag the cursor along its curve. The readout cursor is always on the currently selected curve.
	The mouse pointer is over an object on the graph such as a text label. The mouse can be used to drag the object to a new position on the graph. Double clicking the mouse displays a dialog which allows the object to be edited.

Table 1 Mouse Pointers and their Functions

Graph Main Menu Options

Many graph functions are controlled from the main menu option "Graph" shown in Figure 16.

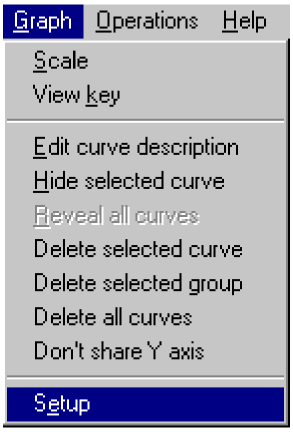


Figure 16 The Graph Main Menu

Each option is described below. Options that are not available at any time are displayed in grey instead of black.

Scale	The scaling dialog is displayed allow axis ranges to be set manually, and autoranging to be configured. See section 0.
View key	A curve key is displayed in a scrolling window. This is an alternative to the curve key displayed below the graph. It is most useful when there are a large number of curves in the graph. It is not printed with the graph.
Edit curve description	See the Curve Popup Menu, section 0.
Hide selected curve	See the Curve Popup Menu, section 0.
Delete selected curve	See the Curve Popup Menu, section 0.
Delete selected group	See the Curve Popup Menu, section 0.
Delete all curves	See the Curve Popup Menu, section 0.
Don't share y axis	See the Curve Popup Menu, section 0.
Setup	The graph set-up dialog is displayed - see section 0.

The Graph Scaling Dialog

The graph scaling dialog is accessed from the "Graph|Scale" main menu option. It is shown in Figure 17.

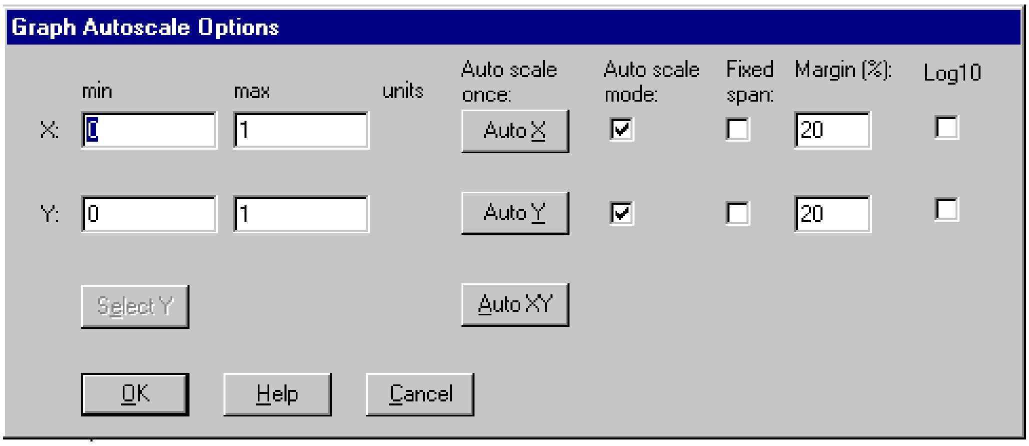


Figure 17 The Graph Scaling Dialog

The ranges of the x and y axis can be entered as required. If there is more than one y axis, click on “Select y” to cycle through them.

You can automatically adjust the range of the x or y axis to match the range of the data (autoscale) actually present by pressing the “Auto x” or “Auto y” button respectively. Both x and y ranges can be autoscaled by pressing the “Auto XY” button.

This dialog also allows autoscaling to be performed automatically when new data is added to the graph. Put a tick by “auto scale mode” for x or y if you want that axis to adjust automatically to accommodate new data points as data is added to the graph. In this mode, an axis range is extended beyond the value of the new data point by a fixed percentage, when can be entered in the “Margin” field for that axis. The margin is used to avoid rapidly repeated rescaling of a graph as each new data point is added.

It is possible for the span of an axis to remain unchanged when it autoscales automatically as new data are added. The axis range is offset instead of extended. Place a tick beneath “Fixed span” if you want to use this feature.

The Graph Setup Dialog

The graph set-up dialog is accessed from the “Graph|Setup” main menu option. It is shown in Figure 18.

You can choose whether to display optional elements of the graph by placing ticks in the dialog. The connecting lines between data points, the cross at the data point, the axis grid and the readout cursor are all optional. The descriptive text, which is not always present, above the graph can also be optionally displayed. If engineering format is selected, all axis exponents are multiples of 3.

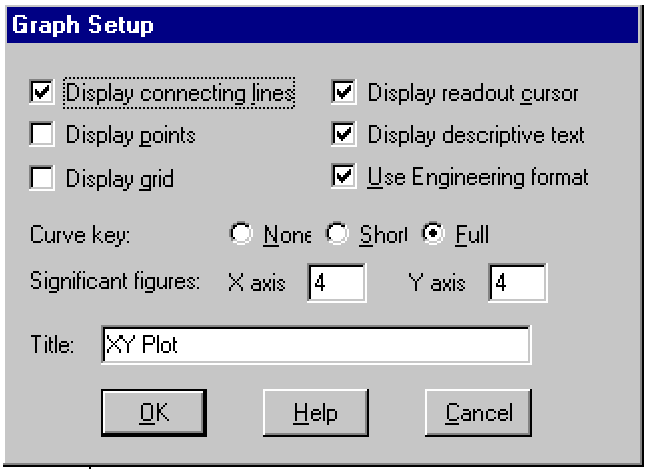


Figure 18 The Graph Setup Dialog

The curve key can appear below the graph. It displays a sample of each curve's line colour, with some descriptive text. If a short key is selected, a number is displayed instead of the descriptive text.

The number of significant figures to display in axis labels can be entered in the "Significant figures" field. If a title is entered, it appears at the top of any graphs printed out.

The Graph Popup Menu

If the right mouse button is pressed when the mouse pointer is over the graph, a popup menu appears as shown in Figure 19.

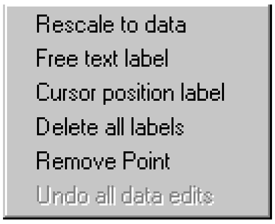


Figure 19 The Graph Popup Menu

Menu options that are not currently available are displayed in grey. Each option is described below.

Rescale to data	Rescale the x and y-axis to match the range of the data displayed exactly.
Free text label	The readout cursor must be moved to a point on a curve before this option is available. This option allows a box containing text to be place on the graph. A dialog box invites you to enter any text you wish. The box is then placed on the graph, and may by positioned by dragging it with the mouse. Note that the text label is always positioned relative to the curve point that was selected when you created it. When the graph's scale is changed and the point's position on the display changes, the text label will also change position. A text label may be moved at any time by dragging it with the mouse. Double click on the label to display an editing dialog, which allows you to change the text, change the label's colour, or delete the label.
Cursor position label	A cursor position label is exactly like a free text label (see above), except that it initially contains the x and y co-ordinates of the readout cursor. Double click on the label to display the editing dialog as above.
Delete all labels	All text and cursor position labels are removed from the graph.
Remove point	This deletes the point selected by the readout cursor.
Undo all data edits	This restores any points deleted by the readout cursor.

The Axis Popup menu

If the right mouse button is pressed when the mouse pointer is over a graph axis (x or y) a popup menu appears as shown in Figure 20. If there is more than one y axis then make your choice of y with the mouse pointer.

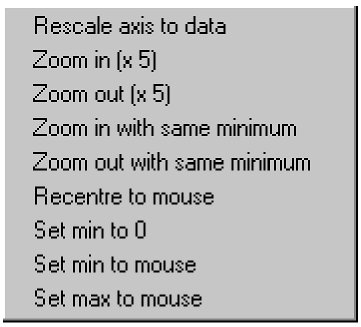


Figure 20 The Scale Popup Menu

Menu options that are not currently available are displayed in grey. Each option is described below.

Rescale axis	Use to data to change the scale to match the data exactly. This is for the chosen axis (x or y) unlike **Rescale to data** in the Graph popup menu, which scales **both** x and y.
Zoom in (x5)	Increases the scale of the chosen axis by a factor 5. The centre point of the axis is unchanged.
Zoom out (x5)	Decreases the scale of the chosen axis by a factor 5. The centre point of the axis is unchanged.
Zoom in with same minimum	As above, but the x or y minimum is fixed before scaling
Zoom out with same minimum	As above, but the x or y minimum is fixed before scaling
Recentre to mouse	Move the axis so that the selected value of x or y is in the centre of the scale
Set min to 0	Shift and rescale the axis as necessary to make its minimum value equal to zero.
Set min to mouse	Rescale the axis so that its minimum axis value is that selected with the mouse pointer.
Set max to mouse	Rescale the axis so that its maximum axis value is that selected with the mouse pointer.

The Curve Popup Menu

If the right mouse button is pressed when the mouse pointer is over a curve key, a popup menu appears as shown in Figure 21. Note that the curve key is only visible when there is at least one curve in the graph and the curve key is not set to “none” in the graph setup dialog (see a later section in this manual).

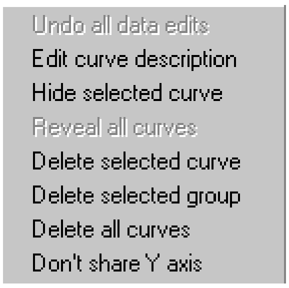


Figure 21 The Curve Popup Menu

Menu options that are not currently available are displayed in grey. Each option is described below.

Undo all data edits	This restores any points deleted by the readout cursor.
Edit curve description	A dialog appears which allows a small amount of text to be entered for the curve key.
Hide selected curve	The currently selected curve and its key are hidden. All hidden curves may be revealed by selecting “Reveal all curves” from this menu.
Reveal all curves	This reveals any hidden curves.
Delete selected curve	The selected curve is deleted.
Delete selected group	All curves in the same group as the selected curve are deleted. A group of curves all share the same x data, can be loaded from, and saved to a single data file.
Delete all curves	All curves in the graph are deleted.
Don’t share y axis	Whenever possible, curves with the same y units share the same y axis. This may be overridden by selecting this menu option.

Copy a Graph to the Windows Clipboard

The contents of a graph may be copied to the Windows clipboard by using the "Edit|Copy" menu option. This allows graphs to be pasted into other Windows programs such as Microsoft Word, for preparation of reports. Graphs are copied to the clipboard in bitmap format.

Printing and Plotting

A graph may be printed or plotted to any printer or plotter supported by Windows. To print a graph, first select the "File|Printer setup" menu option. A list of printers available on your system will be displayed. Select the one you require. You may configure the printer settings, if you wish, by clicking on the Setup button. Otherwise, click on OK. Now you may print the graph by using the "File|Print" menu option. A dialog box will be displayed briefly as data is transferred internally to the Windows print spooler, and then printing will commence. Note that printing at high dot resolutions on laser and matrix printers can take some time, so it is often better to choose a medium or low resolution using the printer set-up button described above. If you wish, you can preview the printout by using the “File|Print preview” menu option.

Graph Colours and Fonts

The default colours and fonts may be changed as described in 0

Menu Item: Basic window

Advanced users may make use of MMS Basic, either directly or by embedding Basic code in a sequence from Measurement Sequencer.

MMS Basic is compatible with Microsoft Visual Basic™ and MMS features can be controlled from Basic.

Clicking the Basic window menu item or equivalent icon opens the Basic script window as below. In this window you may write, debug and run Basic programs and also write dialogs using the dialog editor.

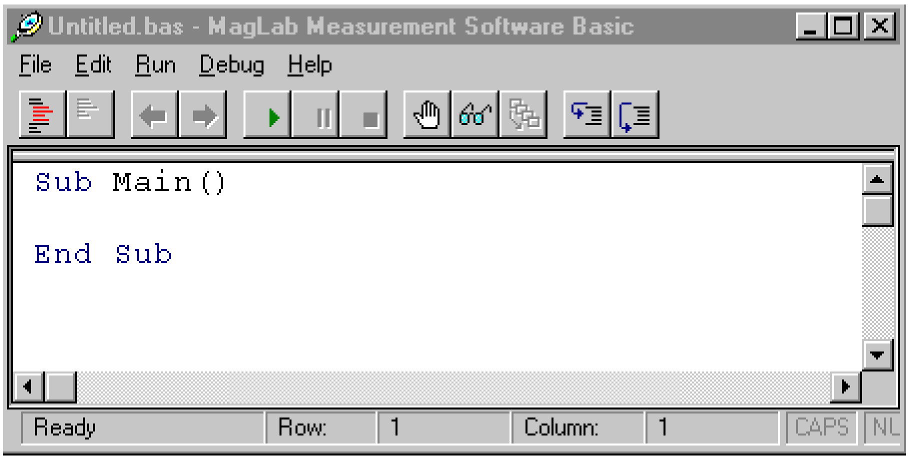
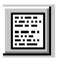


Figure 22 Basic script window

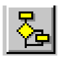
Advice on the writing of Basic code is outside the scope of this document. Please refer to the considerable on-line Help, which is accessible from the Help menu in this window.

Menu Item: Edit file



This is a simple text editor useful for the examination of MMS data files.

Menu Item: Measurement Sequencer



Measurement Sequencer provides a set of high level actions to enable you to write control and measurement sequences for MagLab equipment to suit your own specific requirements. A Sequence is made up of a set of Actions.

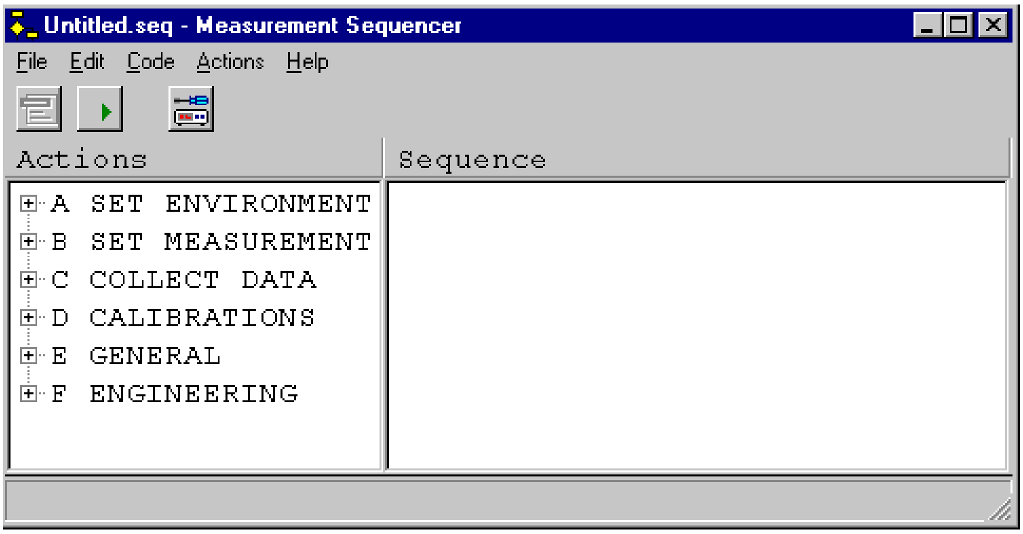


Figure 23 Measurement Sequencer window

Measurement Sequencer Actions relate to the measurements that you wish to make and it is possible to write sequences without programming experience or knowledge of Basic. Basic code, based on MMS Basic, is automatically created from your sequence when you run it.

Measurement Sequencer allows you to write and run sequences on the same computer. This allows you to include multiple real-time graphical displays within a sequence and define new sequence actions specifically suited to your needs. Extensive help on Sequence writing is provided in a separate document "MagLab Measurement Software - Sequencer Manual".

Note: Any of the following actions will pause an active sequence:
a) Placing the cursor on a Plot window and pressing the right mouse button
b) Pressing the "Alt" key
c) Opening (setup) dialog windows of instrument drivers
d) Highlighting/expanding any menu item
e) Pressing the "Pause" button or "Esc" key

Configure menu

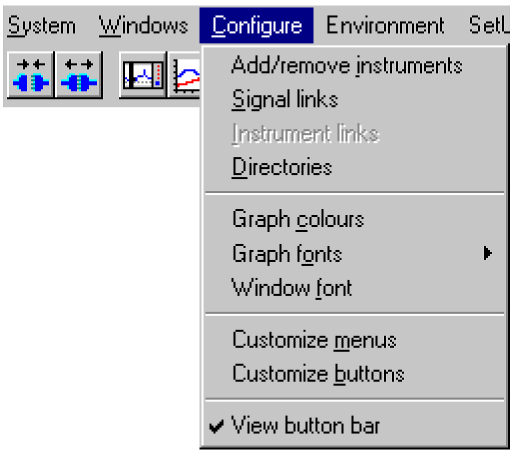


Figure 24 Configure menu

Menu Item: Add/remove instruments

To control an instrument it is necessary to load a corresponding instrument driver. This is accomplished by selecting the "Add/Remove Instruments" option. At this point, a dialog box will be displayed listing any instrument drivers that are already loaded. Figure 25 shows a typical Configure Instruments dialog, in which there is a driver for the ITC503 temperature controller already loaded, with a configuration name of "default".

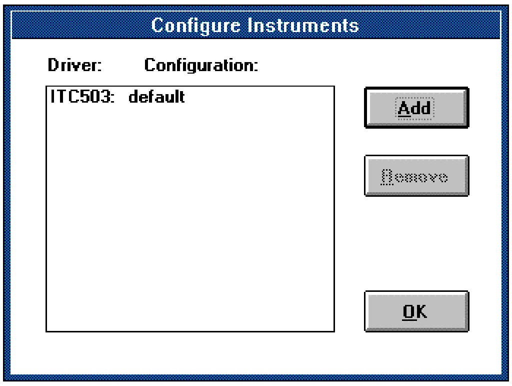


Figure 25 The Configure Instruments Dialog showing Instrument Drivers in use.

To add further instruments, click on the "add" button. A list of available instrument drivers will be displayed. Note that this list may vary from system to system, depending on the drivers actually supplied.

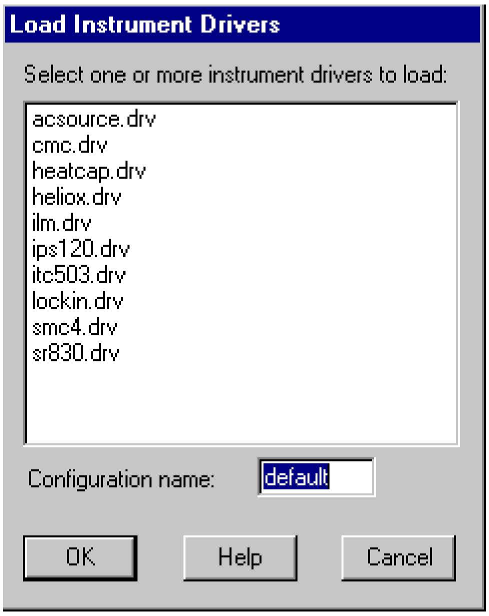


Figure 26 The Load Instrument drivers dialog box, showing a list of instrument drivers available for loading.

Figure 26 shows a **typical list** of instruments. To load an instrument driver, select one from the list using the mouse and click on "OK". Enter a configuration name for the instrument as shown. The configuration should consist of up to 8 characters which are used to distinguish multiple instances of the same instrument driver. In this example, instrument is about to be loaded with a configuration name of "default" - further instances of the same instrument could be loaded subsequently with different configuration names. The configuration is used when creating a file name for storing the instrument's setup. In this example, setup information about an ITC503 would be saved in a file called "default.503" when exiting from MMS, and retrieved from this file when MMS is next run.

Controlling an Instrument from MMS

Once an instrument has been loaded it can be controlled and its signals read in several ways. The simplest method is to control the instrument from its instrument driver, which depends on the specific instrument. This facility will be for each instrument in detail in Appendix 2: Instrument Drivers. Another method is to control the instrument via the Menu Item: Measurement Sequencer. Finally, instrument signals may be displayed graphically using Menu Item: XY plot.

Menu Item: Signal links

A signal link is a connection between an output and an input signal. The output signal must have the updateable property and the input signal must have the writeable property.

When a signal link has been made, the input signal value is set each time the output signal has a new value. For example, if a link is made between an IPS120 magnet controller output (H) and a Heliox instrument input (H), the Heliox is notified of new values of field each time it is measured by the IPS120.

Use ADD in the signal links dialog to add a signal link. The options available will be restricted to those instrument drivers, which have been loaded. Examples of signal links are given in the next Figure.

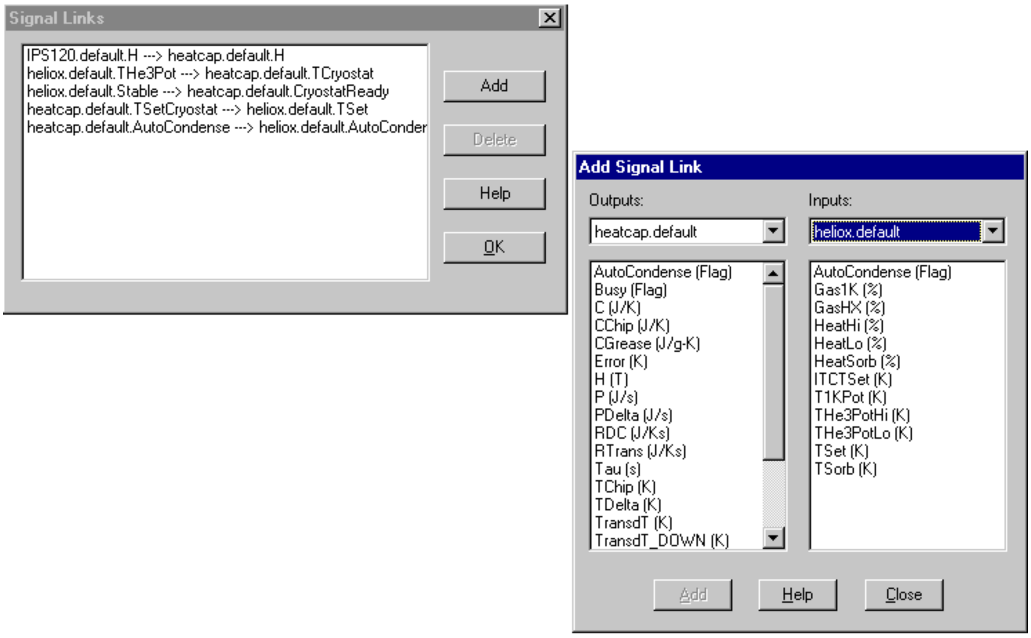


Figure 27 Dialogs for changing signal links

Menu Item: Instrument links
This feature is required only for Oxford Instruments engineers so is greyed out.

Menu Item: Directories
MMS uses various types of file, which are stored in default sub-directories of the MMS data directory. The paths to these sub-directories may be changed by accessing the main menu "Directories" option. This displays a dialog box, which allows you to enter default paths for data files and macro files. Locations of system data files must not be changed so the system data directory cannot be changed. The SetUp for MagLab experimental probes create sub-directories appropriate to the measurement within the main mmsdata\expt\ subdirectory.

Menu Item: Graph colours and fonts
All graphs in MMS share the same colours and use the same fonts for text. These colours and fonts can be changed by using the main menu items “Configure|Graph colours” and “Configure|Graph fonts”. The colour and font used for each part of the graph can be chosen. These settings will be retained for future use by MMS.

Menu Item: Window font
This can be used to alter the text appearance inside the Measurement Sequencer, Basic window and Edit file window.

Menu Item: Customise menus
You may use this utility to create your own menu. This can be useful for collecting together a set of custom sequences or Basic programs. Selection of the menu item causes the sequence or Basic program to run.
The Customise Menus dialog for your system will resemble the example below.

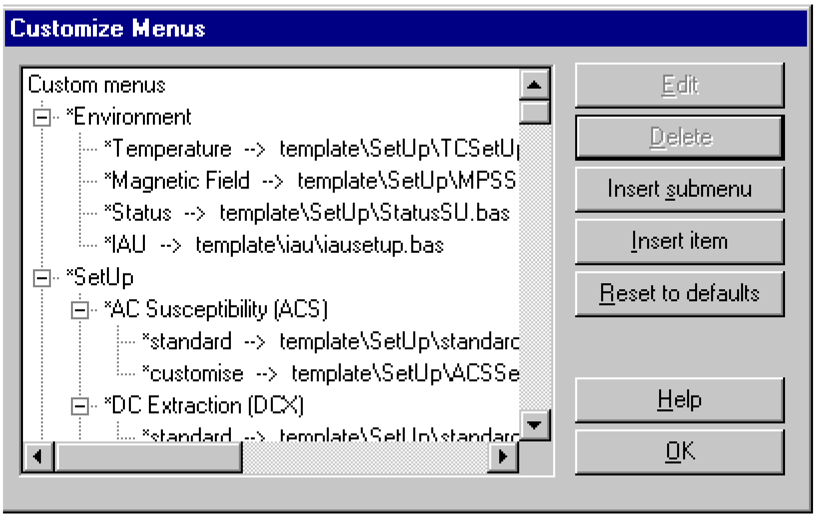


Figure 28 The customizing menus dialog

The details will depend on which experimental probes (if any) have been selected at the time of Installing MMS. All standard menu items have text preceded by an asterisk *.

MMS cannot prevent you removing these menu items if you wish to do so.

Click [Reset to defaults] at any time to cancel all changes (including deletions of standard menu items) and restore the original menu structure.

Click on [Insert submenu] to create a new submenu at the top of the list, as shown in the next dialog. The new submenu will appear to the left of the Environment menu in the MMS menu bar.
To position the new submenu elsewhere, first select a standard submenu item (for example, SetUp) before inserting the new submenu. This will then appear to the left of the selected submenu (SetUp).

Use [Edit] to edit the text of any menu item.

Use [Insert item] to insert individual menu items. When these are selected, followed by [Edit] the next dialog is displayed.

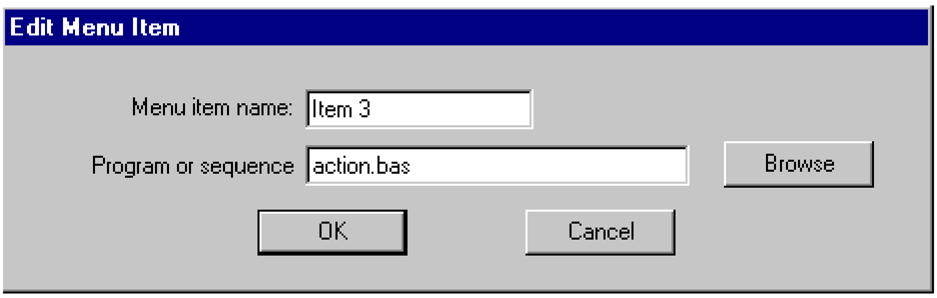


Figure 29 The Edit Menu Item dialog

Use this to edit the text and select a sequence (*.seq) or Basic file (*.bas) which is to be run when the menu item is selected. You can create a list of the type shown at the top of the next dialog.

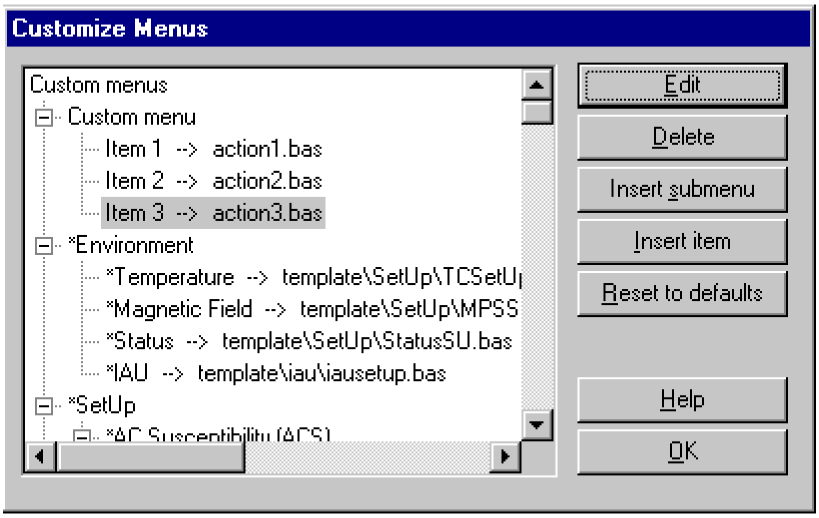


Figure 30 Customize menus dialog showing new items being added

Click OK to return to the main MMS workspace, where the new menu will now appear.

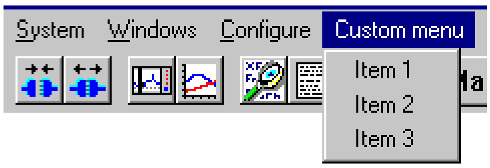


Figure 31 The newly created custom menu and its items

Menu Item: Customise buttons

You may find one or more Customised buttons created when you are Installing MMS as shown in the Customise Buttons dialog below.

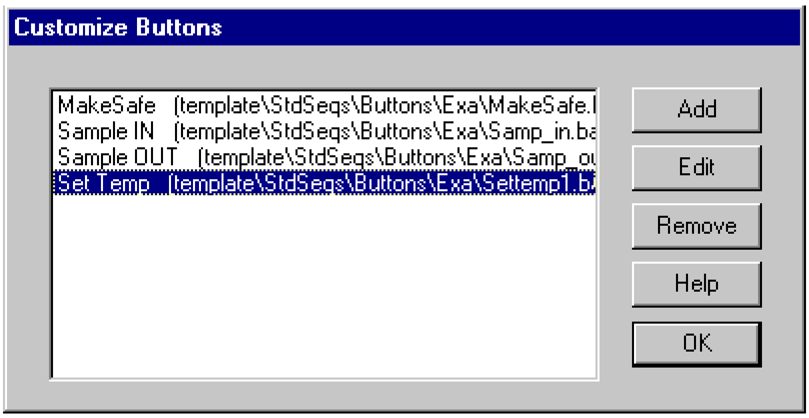


Figure 32 The Customize Buttons menu

MMS cannot prevent you removing these menu items if you wish to do so.

Notice that these buttons run Basic files found in the \StdSeqs\Buttons subdirectory. Use [Add] and [Edit] to create the button text and select the sequence or Basic file.

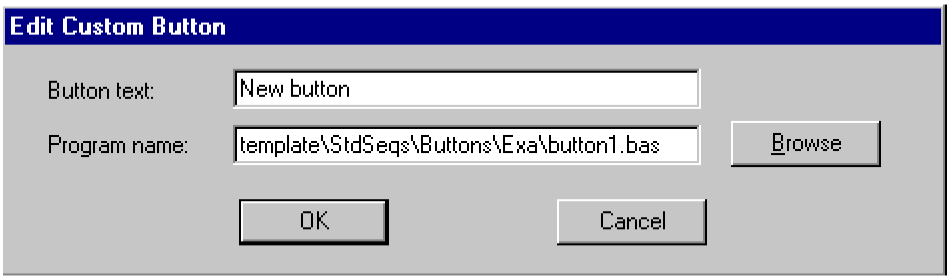


Figure 33 The Edit Custom Button dialog

When you click OK the new button appears on the button bar as shown below.

Figure 34 Menu bar showing some added custom buttons

Menu Item: View button bar

This is normally active. If View button bar is de-selected then the button bar (including Custom buttons) will not be visible, giving greater workspace.

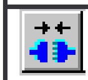
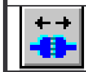
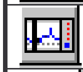
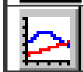
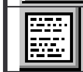
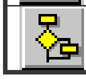
Help menu

Figure 35 The Help menu

This menu provides on-line Help for ObjectBench, from which MMS is derived. Use Contents to view an introduction to main software functions. "Search for help on...." provides a searchable alphabetical index of topics. Function key F1 will provide context sensitive help on the active window.

MMS Button bar

There are 7 non-configurable buttons, each with an equivalent menu item. Please consult the section for the menu item for further details.

Button icon	Equivalent menu item
	Menu Item: Connect Al
	Menu Item: Disconnect All
	Menu Item: XY plot
	Menu Item: Graphical Analysis
	Menu Item: Basic window
	Menu Item: Edit file
	Menu Item: Measurement Sequencer

There are 4 configurable buttons listed below. Each runs a Basic program to perform the given function.
For information on how to create your own buttons refer to Menu Item: Customise buttons.

Tutorials

Log Temperature controller using XY plot
Experimental measurements commonly require control of the sample temperature. It is very useful to be able to monitor this in a simple way. MMS provides this facility using the XY plot function.

This example supposes that a sample has been fitted to a measurement platform and its temperature set to a chosen value. We wish to monitor its temperature and, when the experimenter decides that it has reached a stable value, to begin data collection.

Click on the menu item or icon for XY Plot.
In the Inputs menu

- Click Trigger. Choose the trigger option, say Time, 5 seconds.
- Click x signal. Choose the x signal, in this case Time
- Click y signals. Choose the y signals. In this example we have chosen Tset (the set temperature) T1 (the control temperature, T2 (representing the sample temperature) and the heater voltage.

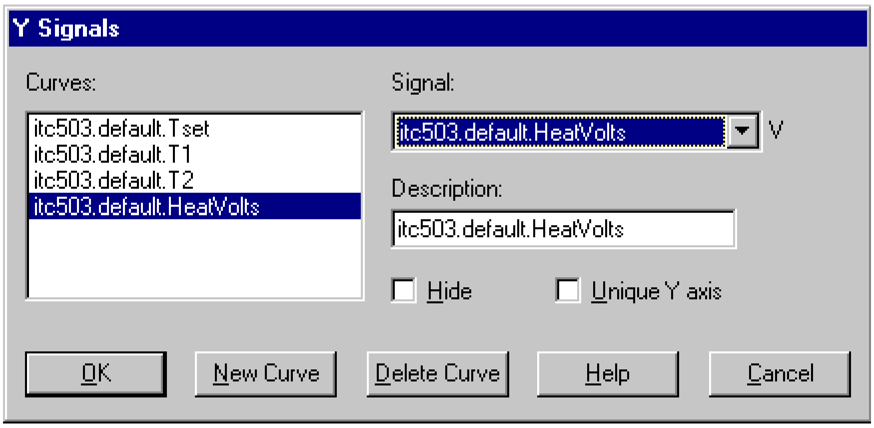


Figure 36 Setting up XY plot Y-signals

- Click Connect to connect the signals to the graph.

The graph will now start recording Tset T1, T2 and HeatVolts as a function of time, with a set of points appearing every 5 seconds. After about 1000 values of each parameter (= 5000 seconds) additional points will cause earlier ones to be deleted. If you wish to record continuously and/or retain a record you should click File, Log and choose a filename in which to Save the data.

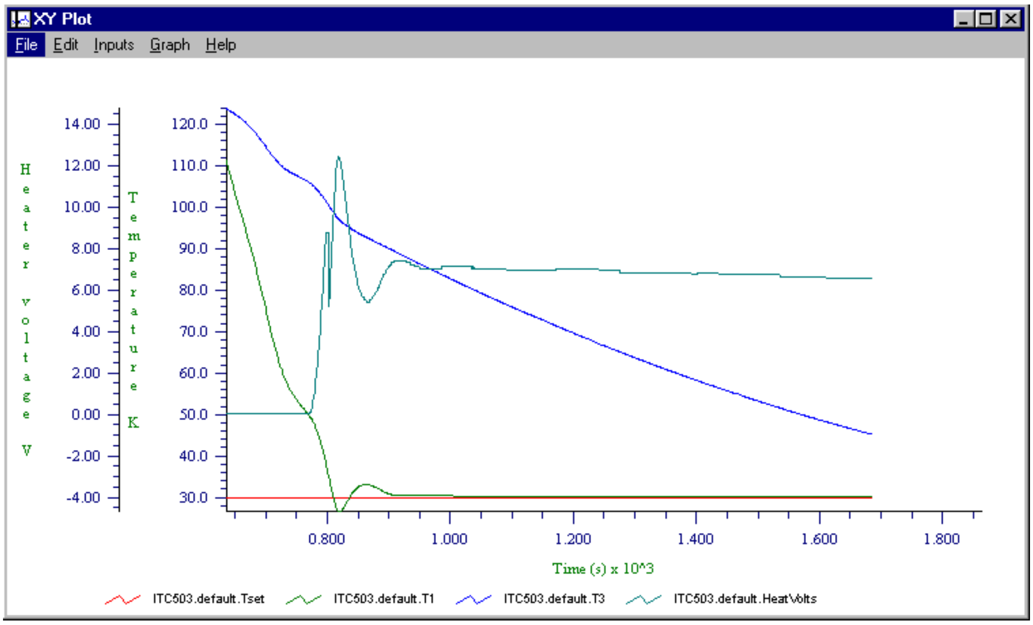


Figure 37 Example XY plot, plotting ITC signals vs time

There is no reason to Disconnect the XY plot when the experiment is begun since the plot can be minimised and allowed to run continuously in the background. As required you can clear old data from the graph (Graph, Clear data) and re-start the graph x-axis (Graph, Reset time).

Set up and control a small group of ISOBUS instruments

Oxford Instruments ISOBUS allows communication along a single serial RS232 line with up to 8 instruments (such as temperature controllers and magnet power supplies). A special ISOBUS cable is required. For more general information about ISOBUS refer to the appropriate section in the manuals for the individual instruments. In this example we take two temperature controllers (itc503), cryogenic level meter (ilm211) and magnet power supply (ips120).

Click on Configure, Add/remove instruments and Add each instrument in turn. Since there are two temperature controllers, you must give each controller a unique configuration name. The final selection might be

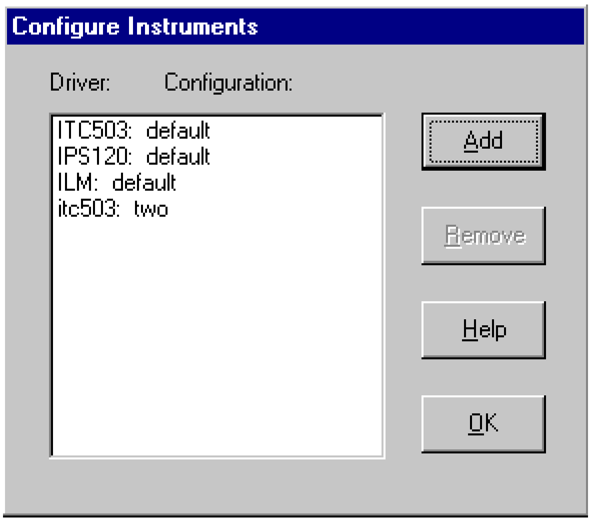


Figure 38 Configuring instruments dialog. Use the 'Add' button to add instruments

Where the second temperature controller has been configured as ITC503.two
Each instrument driver and instrument must then be given a unique ISOBUS address (an integer from 1 - 8). Do this using each instrument drivers' Config menu.

There are other settings and parameters which will be unique to each instrument, and so you may need to examine and edit the contents of the Setup menu. For a full discussion on SetUp options please refer to the individual instruments' manual. When you Connect each instrument (or Connect all) you will then be in a position to control each instrument using the (virtual) instrument driver.

Using third party instruments

The following notes exemplify the incorporation of commands to third party instruments into the MMS sequencer.

Click the Edit Custom Actions button in the Sequencer button bar.

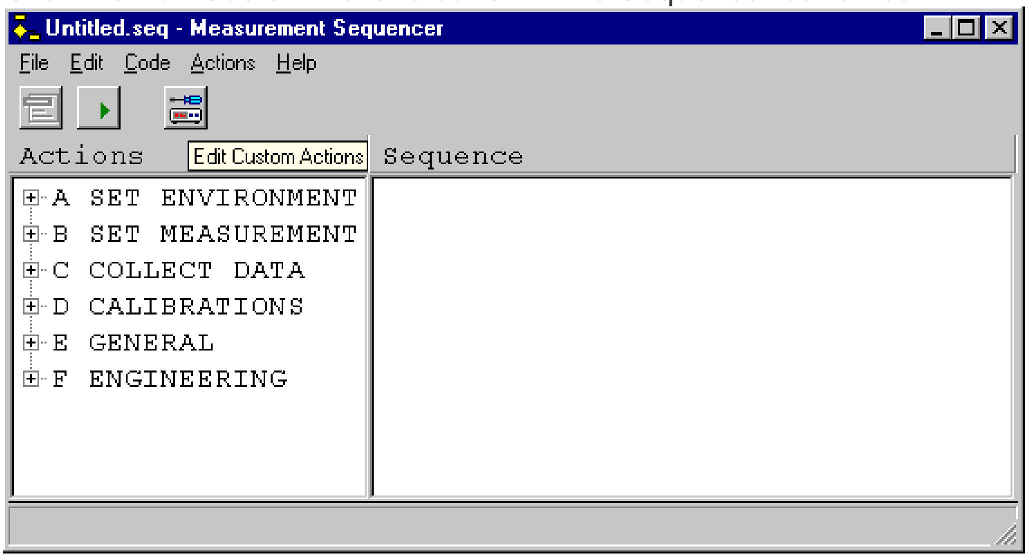


Figure 39 Main Measurement sequencer window

Then select Create Action in the Create/Edit dialog and proceed to follow the instructions in the New Action Engineer wizard.

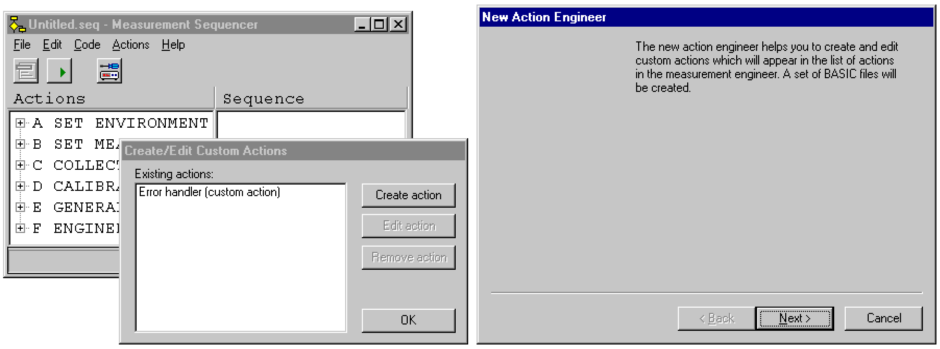


Figure 40 Creating a new custom action

In the next dialog select Instrument Action (to send a command to the TSX).

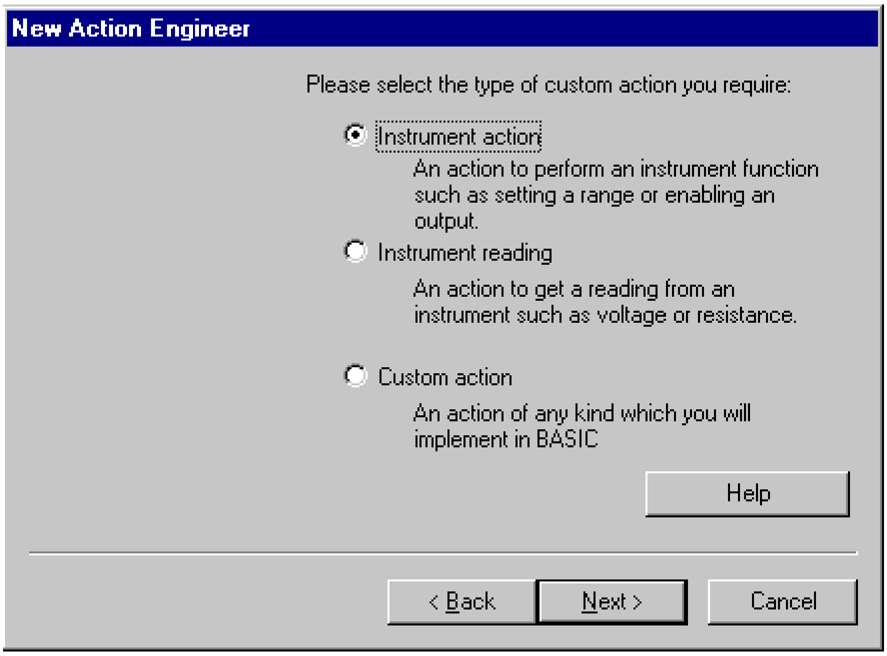


Figure 41 Selecting the type of action

You will be asked to give a name to the instrument and give information about its interface. The example dialog following assumes the GPIB has been chosen and the GPIB instrument address identified. The terminator character for the TSX is (ASCII)10.

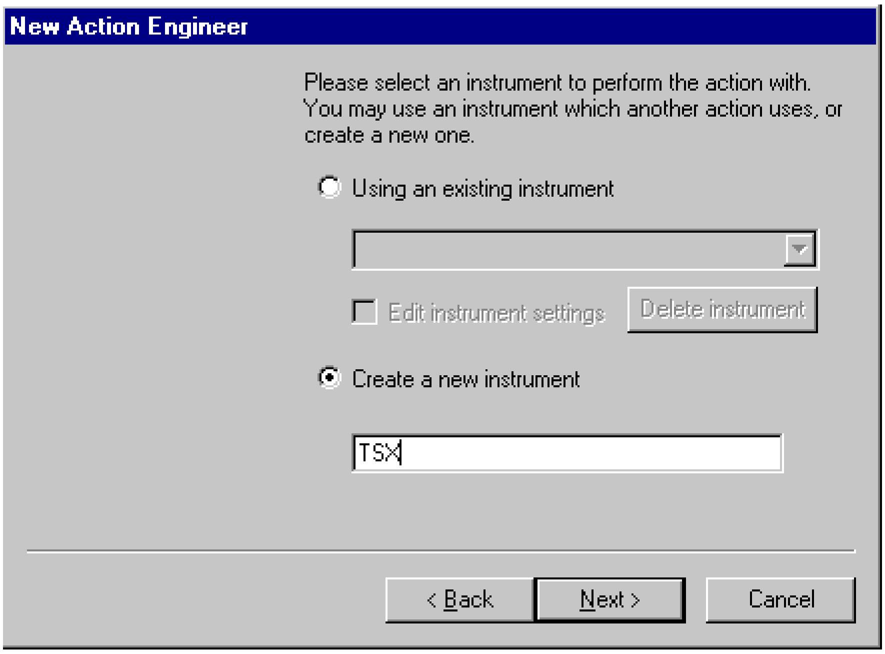


Figure 42 Creating a new instrument called "TSX"

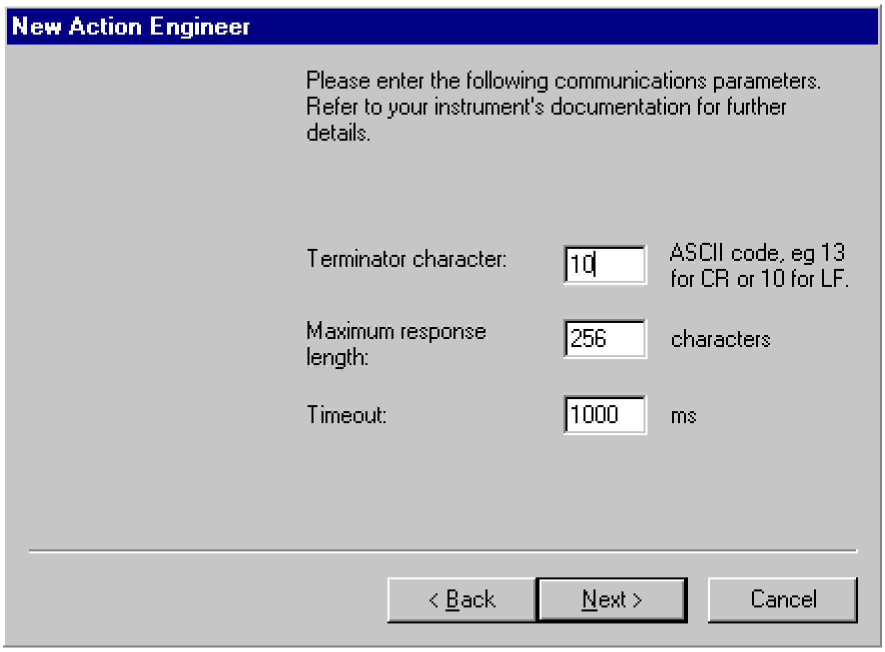


Figure 43 Setting the communication protocols for TSX

When you have completed the remaining steps you will find that the Sequencer now includes a TSX command action. Use this to enter any command using the TSX syntax.

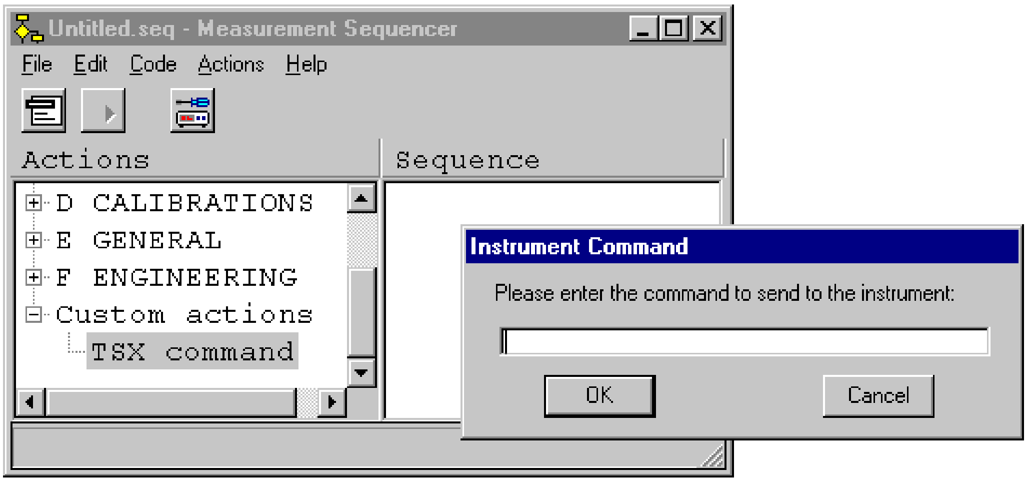


Figure 44 Defining a new command for the new instrument

Our example in the next dialog shows "OP 1", which switches the TSX output ON.

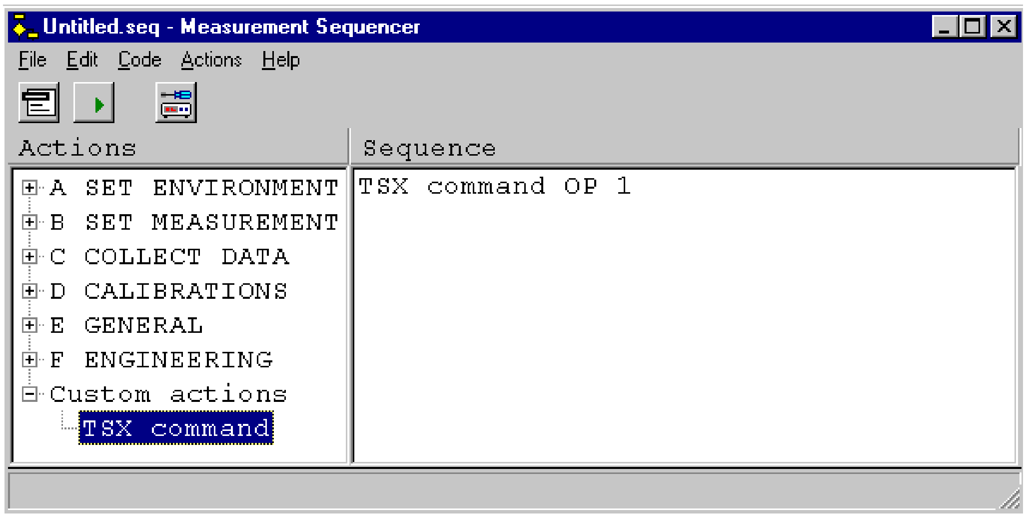


Figure 45 Main measurement sequencer window showing added instrument command

In order to interrogate the instrument (take a reading from it) you must return to the New Action Engineer and select "Instrument reading" as below.

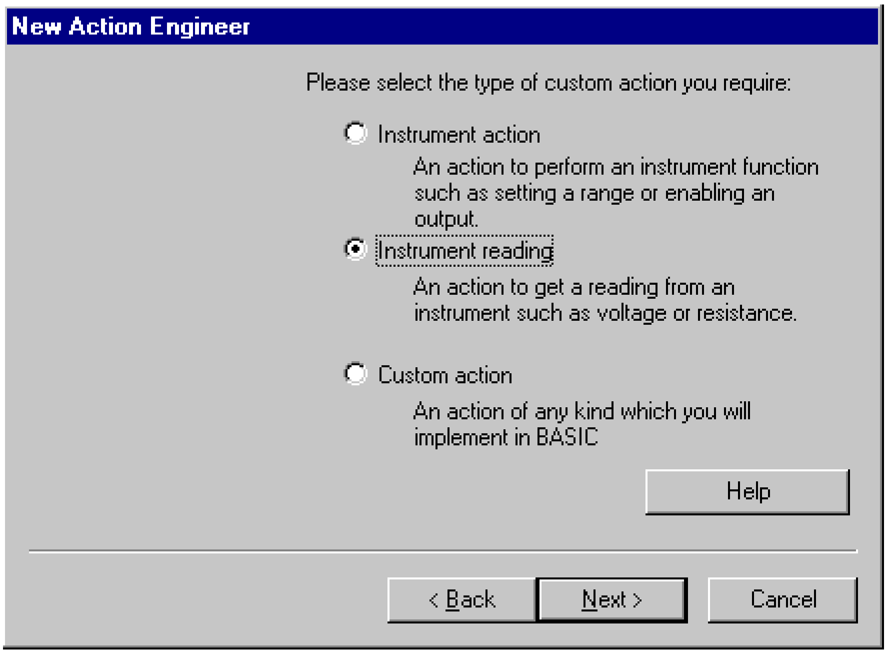


Figure 46 Adding an instrument reading

Proceed to select the instrument.

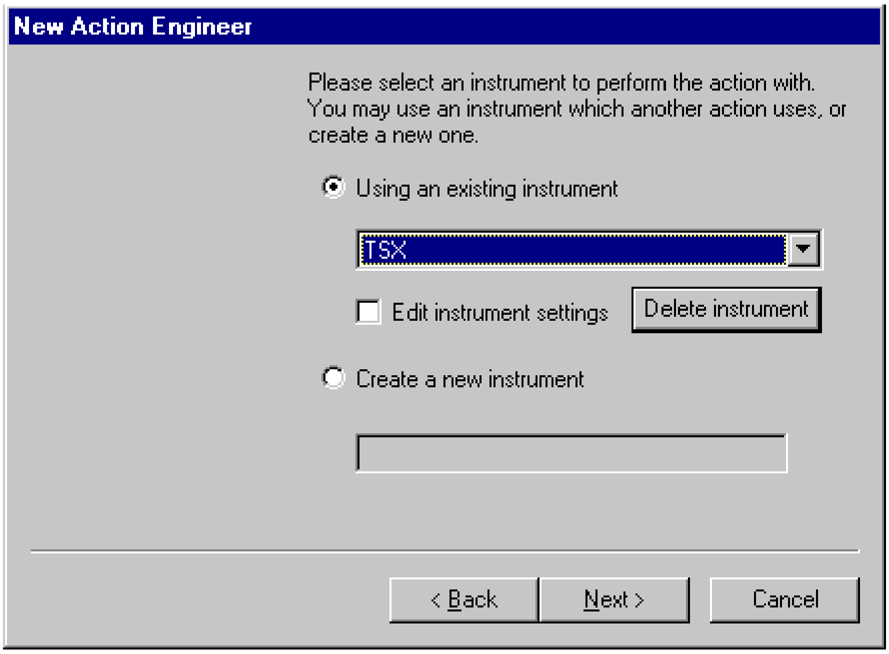


Figure 47 Adding the function to an existing instrument

As we have already defined the TSX it appears in the drop down list of existing instruments. Proceed to set up information about the instrument signal. Our example uses the TSX current, read using the **IO?** command (see TSX manual). The response string from the instrument begins with the numerical argument so the "position" is entered as 1.

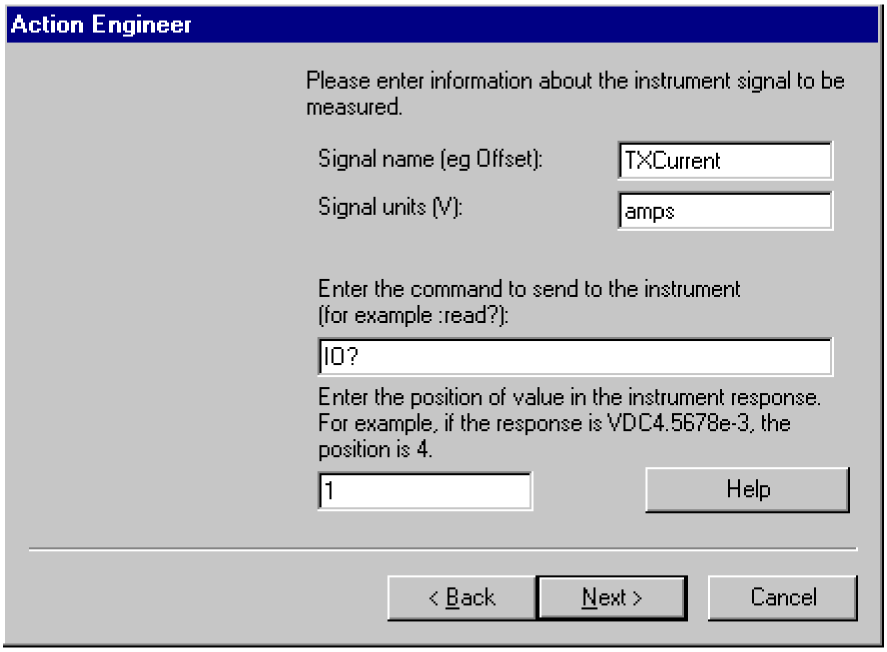


Figure 48 Defining the instrument reading

The defined signal name will then be available in the list provided by the Measure action as below.

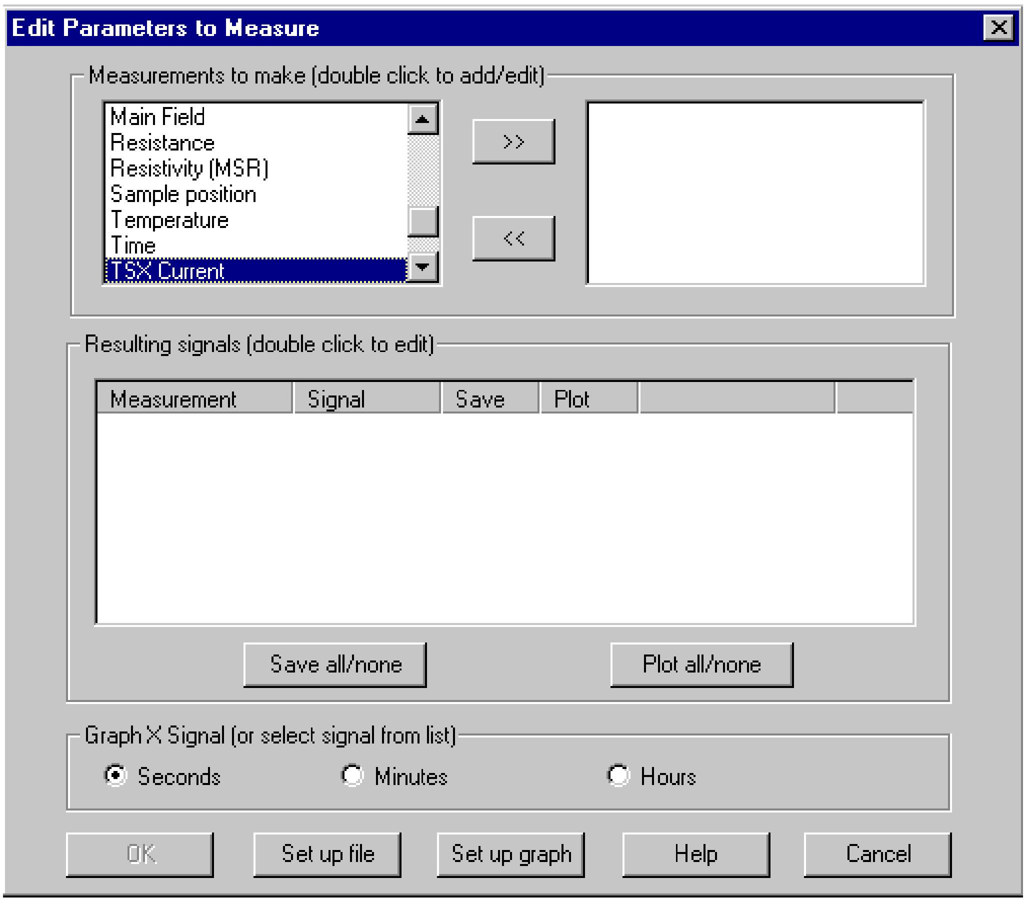


Figure 49 The Measurement window showing the new signal

It is beyond the scope of this document to discuss writing sequences in detail. For this please refer to your Sequencer manual. We would expect that a simple sequence that increments the TSX current by 0.1 amp steps to resemble the following.

```
Remark ' Prompt for Variables
Prompt for 4 Variables

Remark ' Initialise the instruments
TSX command *RST

Remark ' Set Initial Conditions
TSX command DELTAI0.1

Remark ' Make the  Measurement Scan
Reset time to zero
TSX command OP1

GENERAL LOOP [counter] times
  delay [Delay] seconds
  Measure Time, Temperature, TSX Current
End GENERAL LOOP

TSX command OP0
```

Customer Support

Warranty

The Oxford Instruments customer support warranty is available to all our customers during the first 12 months of ownership from date of delivery. This warranty provides:

- Repair to faults that are a result of manufacturing defects at Oxford Instruments.

Technical support

To obtain technical support you will need to quote your Oxford Instruments order number. Please contact your nearest Customer Support centre as follows:

Europe, RoW, Main Office
Tel: +44 (0)1865 393 311
E-mail: helpdesk.nanoscience@oxinst.co.uk
Web: www.oxford-instruments.com

USA Office
Tel: +1 800 447 4717
E-mail: csg@ma.oxinst.com
Web: www.oxford-instruments.com

Japan Office
Tel: +81 03 5245 3261
E-mail: oikkcsri@oxford-instruments.ne.jp
Web: www.oxford-instruments.com

China Office
Tel: +86 21 63608530/1/2/3
E-mail: info@oxford-instruments.com.cn
Web: www.oxford-instruments.com

Additional services available from Oxford Instruments

Oxford Instruments Direct – provides one-stop shopping for cryogenics, magnetic, vacuum and associated laboratory products as well as hard to find reference for low temperature physics, optics, thermometry and laboratory safety practices. You can also visit on-line at www.oxinstdirect.com. Ordering parts is easy via phone, fax or email. Online ordering is available in North America.

About Oxford Instruments

Oxford Instruments specialises in the design, manufacture and support of high-technology tools and systems for industry, research, education, space, energy, defence and healthcare.

We combine core technologies in areas such as low temperature and high magnetic field environments; X-ray, electron and optical based metrology; nuclear magnetic resonance, advanced growth, deposition and etching. Our aim is to be the leading provider of tools and systems for the emerging nanotechnology and bioscience markets.

Oxford Instruments NanoScience Limited

Oxford Instruments NanoScience creates high performance environments for low temperature and high magnetic field applications in physical science research and process development down to the atomic scale.

The business has a strong capability in advanced cryogenics and applied superconductivity. Through the application of these technologies we deliver solutions that meet the exacting needs of scientists working at the forefront of fundamental physics, applied physics, materials science and next generation device development. With an extensive customer network, a strong reputation for performance and quality, we value the support we have provided to world leading research scientists in their pursuit of excellence.

Appendix 1: Data file format

All MMS data files created by macros and XY plotting windows may be read in by an analysis window. These are ASCII text files in the format described below. This information is useful if you wish to import data from a different ASCII file format by modifying it with a text editor.

```
Tue Jun 19 15:03:57 2001
ferrite at 2.5K
        Time,            H,       Moment,      M(Mass), Tsample(Aver)
           s,            T,          emu,        emu/g,             K

         6:0          6:0          6:0          6:0          6:0
3.240234E+00 0.000000E+00 9.977645E-01 3.464460E+01 2.044000E+00
5.383008E+01 2.500000E-01 -2.315390E+01 -8.039549E+02 2.033000E+00
8.183984E+01 5.000000E-01 -2.484894E+01 -8.628103E+02 2.036000E+00
1.038594E+02 7.500000E-01 -2.542828E+01 -8.829264E+02 2.050000E+00
1.261094E+02 1.000000E+00 -2.577939E+01 -8.951176E+02 2.048000E+00
1.481895E+02 1.250000E+00 -2.605619E+01 -9.047287E+02 2.045000E+00
1.709805E+02 1.500000E+00 -2.626799E+01 -9.120829E+02 2.045000E+00
1.930098E+02 1.750000E+00 -2.644789E+01 -9.183296E+02 2.045000E+00
2.152500E+02 2.000000E+00 -2.661391E+01 -9.240941E+02 2.054000E+00
2.380508E+02 2.250000E+00 -2.676126E+01 -9.292104E+02 2.046000E+00
2.611094E+02 2.500000E+00 -2.689373E+01 -9.338100E+02 2.048000E+00
2.832500E+02 2.750000E+00 -2.701382E+01 -9.379798E+02 2.050000E+00
3.062598E+02 3.000000E+00 -2.712570E+01 -9.418648E+02 2.050000E+00
3.283398E+02 3.250000E+00 -2.720356E+01 -9.445682E+02 2.051000E+00
3.505293E+02 3.500000E+00 -2.730737E+01 -9.481727E+02 2.048000E+00
3.726094E+02 3.750000E+00 -2.739306E+01 -9.511480E+02 2.049000E+00
3.954102E+02 4.000000E+00 -2.744301E+01 -9.528824E+02 2.056000E+00
4.177109E+02 4.250000E+00 -2.751948E+01 -9.555375E+02 2.044000E+00
4.398398E+02 4.500000E+00 -2.756155E+01 -9.569981E+02 2.051000E+00
```

Figure 50 Structure of MMS data files

The first 6 (sometimes 8) lines are reserved for special information. The remainder of the file contains a number of columns of numbers as shown, and may be of any length required. There may be any number of columns, although if the data is to be plotted on a graph there must obviously be at least two columns.

In the above example: -
Line 1 is the date, which is automatically added by MMS when the file is created.
Line 2 is descriptive text entered by the user, of any length.
Line 3 consists of a list of signal names, separated by commas.
Line 4 consists of a list of units names, separated by commas.
Line 5 is reserved, it should be left blank.
Line 6 consists of a list of format codes matching P:Q as shown. P is the number of significant figures in the data. Q is not used and should be set to zero.

Comment lines may be included at any point in the first six lines by preceding them with a ";" character. Note that the number of entries in third and fourth lines must be the same as the number of columns of data. If this is not true, MMS will not load the data file.

Appendix 2: Instrument Drivers

This appendix describes the most common instrument drivers. Instrument drivers are supplied in separate files with extension ".drv", contained within the main MMS program. There are two kinds of instrument drivers, real and virtual. Real instrument drivers are most common and correspond to a physical instrument, such as a temperature controller or power supply. Virtual instrument drivers do not control real instruments, but acquire their inputs from real instrument drivers. Typically, they combine the inputs from the real instrument drivers to perform calculations which help to interpret the results of an experiment. An example of a virtual instrument driver is the Heliox driver.

In general, a complete set of instrument drivers will be supplied with MMS irrespective of your immediate requirements but some of these may be hidden. The method of hiding the driver is a change to its file extension from *.drv to *.dll so that the 'Add Instrument' routine would not find the driver. If you cannot find the driver for your instrument on the "Load Instrument Driver" list (menu route 'Configure -> Add/remove instruments -> Add') then close MMS, use Explorer to navigate to C:\Program Files\MMS (or wherever you installed MMS) and look for a dll file which matches your instrument. For example, add the ITC502 driver to the available list, rename **itc502.dll** to **itc502.drv**. Now restart MMS and navigate back to "Load Instrument Driver" and the itc502 should appear on the available drivers list.

Interfacing Instruments to MMS

Interfacing to the instrument can be via RS232, ISOBUS, GPIB or GPIB Gateway interface. Note that the GPIB interface is optional and will not have been fitted to the instrument unless it was specifically ordered.

The RS232 interface is a standard serial connection that may be used to connect a single instrument to one serial port (or USB port with a suitable converter) on the PC. The ISOBUS interface allows more than one instrument to be connected to a single serial port (or USB port with a suitable converter) on the PC, using a special lead supplied by Oxford Instruments. The GPIB is a standard parallel interface that allows a number of instruments to be connected to a single GPIB adapter card in the PC using a standard GPIB cable. The GPIB Gateway interface allows a single GPIB instrument to act as an ISOBUS master. The alternative methods of interfacing are illustrated in Figure 51 and described more fully in the instrument's manual. Choose the scheme which is most suited to your needs and connect the PC and instrument accordingly.

If you are using a serial port on your PC it will probably be necessary to use a 9 to 25 pin converter. See your computer's documentation for further details of its COM ports.

Having physically connected the instrument to the PC, you should now configure the instrument's interface in software. Select the instrument menu item "Config"/"Interface" to display the dialog shown in Figure 52.

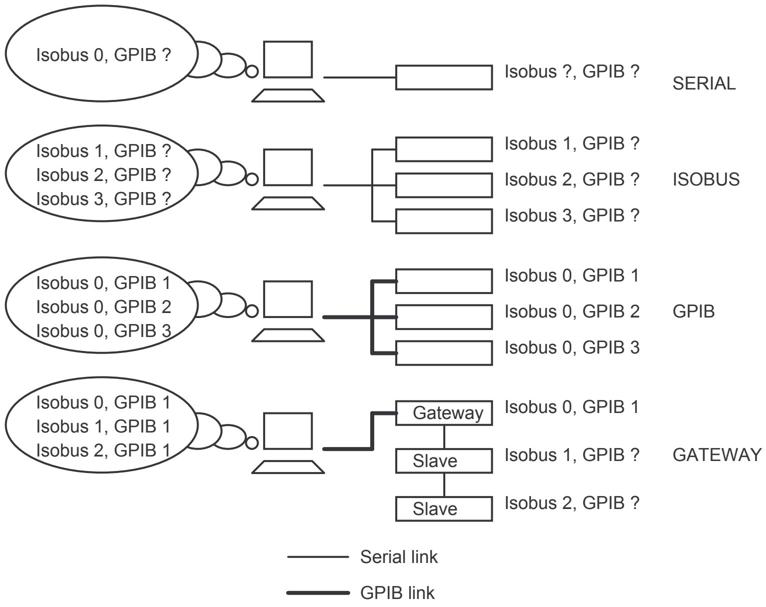


Figure 51 Alternative interfacing methods.

Click next to the interfacing option you require on the left of the dialog: select ISOBUS, GPIB or Gateway. If you require a simple RS232 interface select ISOBUS. Then enter the PC's COM port and the instrument's ISOBUS and GPIB addresses that are to be used in the remaining fields in the dialog. If you have selected ISOBUS or GPIB Gateway, you should now set the instrument's ISOBUS address by clicking on the "Set address" button.

It is important that no two instruments on the same ISOBUS or GPIB bus share the same address, or communication will be impossible.

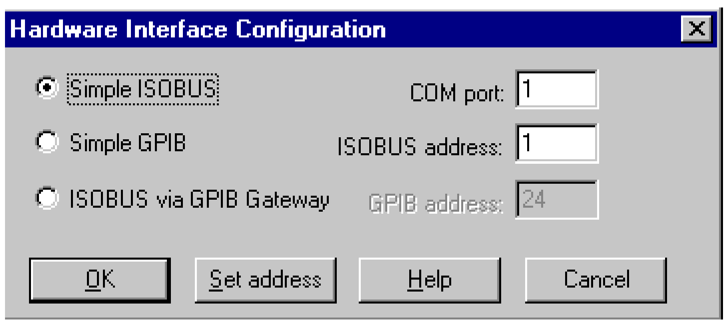


Figure 52 The interface configuration dialog box.

ITC503 Temperature Controller

The Intelligent Temperature Controller model 503 (ITC503) is a one (optionally three) channel temperature controller and monitor designed specifically for cryogenic system control. Its MMS instrument driver allows full remote control, including control of set point, PID control, and ramping. It also allows custom sensor ranges to be designed and downloaded to the instrument. Full details of the instrument are included in a separate manual. The name of the MMS driver is "itc503.drv".

Preparing to Use the ITC503 from MMS

Figure 53 shows the ITC503 instrument driver main window, which is displayed by double clicking on the ITC503 icon.

Before the software front panel can acquire data from the hardware instrument, you must connect (interface) the instrument to the PC and configure the instrument driver correctly as described above.

Select the instrument menu item 'Config -> Instrument to the display the dialog box shown in Figure 54.

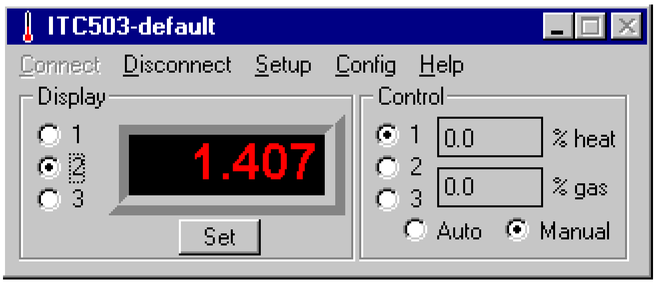


Figure 53 The ITC503 software front panel.

The fields in the ITC503 Instrument Configuration dialog allow you to specify the maximum heater voltage for each channel. The maximum heater volts is the maximum voltage that may be applied to the heater when controlling using a particular channel, and should be set to the maximum safe value permitted for your system. **Note that setting the maximum heater volts too high can damage your system.** If in doubt, contact Oxford Instruments. When the ITC is controlling it will display the applied heat as a percentage of this voltage. So, if the maximum voltage is set at 10V, then 100% output means 10V is being applied across the heater and 50% output means 5V is being applied across the heater. Note: Percentage output does **not** scale as power.

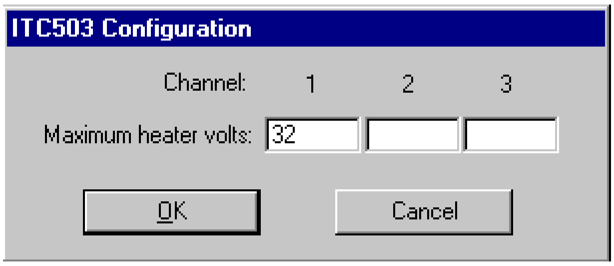


Figure 54 The ITC503 configuration dialog.

Finally, you may "connect" to the ITC503 by clicking on the ITC503's "Connect" menu option. The current temperature should appear on the display and be updated in real time. If this is not the case, an error message will be displayed which should give an indication of the nature of problem. The most likely cause of difficulty is in the interfacing options - if in doubt, review these first.

Operating the ITC503 from MMS
The ITC503 software control panel is shown in Figure 53. It is very similar to the physical instrument front panel and operates in a similar way.

The temperature channel to display is chosen by clicking on the channel 1, 2 or 3 radio button in the "Display" box. The control channel is selected by clicking on the radio button labelled 1, 2 or 3 in the "Heater" box. Automatic or manual temperature control is selected by clicking on the Auto or Manual radio buttons in the "Heater" box. The current heater output is always displayed in percent.

While one of the ITC503's front panel buttons is pressed by a user, the ITC503 delays responding to data requests by the PC. If the button continues to be pressed, MMS will time out after 5 seconds.

To change the set point, the manual heater output or the PID settings, click on the Set button to reveal the dialog shown in Figure 55

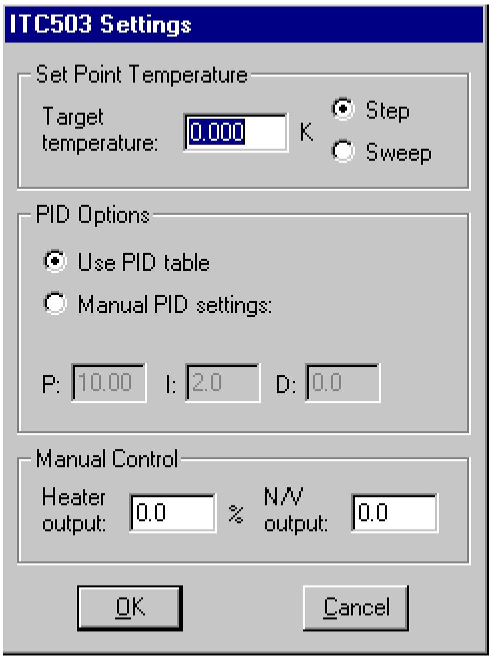


Figure 55 The ITC503 Settings Dialog.

The temperature set point may be entered in the first field of the dialog. When the ITC503 is in manual control mode, the heater and needle valve outputs may be entered in %.

There are two options for the PID settings, "Use PID Table" or "Manual PID Settings".

The "Manual PID Settings" option allows P, I and D settings to be entered manually in their respective fields (see the ITC503 manual for information on how to choose values for P, I and D manually). The new values will take effect at once.

The "Use PID Table" option should be checked if you wish to use a PID vs. temperature table as described below in section 0. This feature allows P, I and D to be determined from a table of temperature ranges and PID settings, and updated automatically whenever a new temperature set point is entered.

The set point may be stepped or ramped from the existing set point at a constant ramp rate. Select one of these options by clicking on the "Step" or "Ramp" radio button. When "Ramp" mode is selected, the temperature field displays the target of the ramp; otherwise, in "Step" mode, it displays the current temperature set point. The constant ramp rate is chosen in the general set-up dialog (see below).

When you have finished using the ITC503 from MMS, click on the Disconnect main menu option to stop communication with the instrument.

The ITC503 General Setup Dialog

The General Setup dialog is accessed by selecting the instrument's "Setup"/"General Setup" menu option and displays the dialog shown in Figure 56

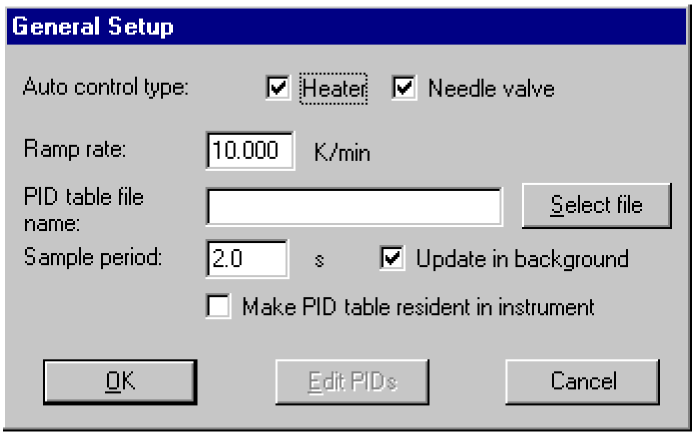


Figure 56 The ITC503 general setup dialog.

This dialog allows you to specify the control type, that is heater, needle valve, or both. Check the appropriate boxes in the dialog - note that you must check at least one box.

The sampling period is the rate at which new readings are obtained from the instrument and the display updated. A typical value is 2 seconds.

It is possible for the ITC503 to use a table of PID values and temperature ranges supplied by the user, so that whenever the temperature set point is changed, suitable PID values from the table are set up. This information is contained in an MMS format data file which is named in the "PID file name" field. The file specification should include the full path, as shown. To select an existing PID file, click on the "Select File" button and choose the file from the list presented. If you wish to create a new PID file, just type its name in this field and edit it. To edit the PID file, click on the "Edit PIDs" button. The PID table selected will not actually be used until the "Use PID Table" option is selected in the ITC503 Settings dialog.

If the "Make PID table resident in instrument" option is selected, the PID table will be downloaded to the ITC503 from the computer and its "Auto PID" mode used to select P, I and D settings for a given set point temperature. Otherwise, the PID table is held in the computer, which updates P, I and D whenever a new set point is entered through the computer. You may wish to make the PID table resident in the instrument if you wish to use it separately from the computer.

The ramp rate used in ramp mode is entered in Kelvin per minute in the appropriate field in this dialog. Ramp mode is selected in the settings dialog described above.

The ITC503 RAM Setup Dialog

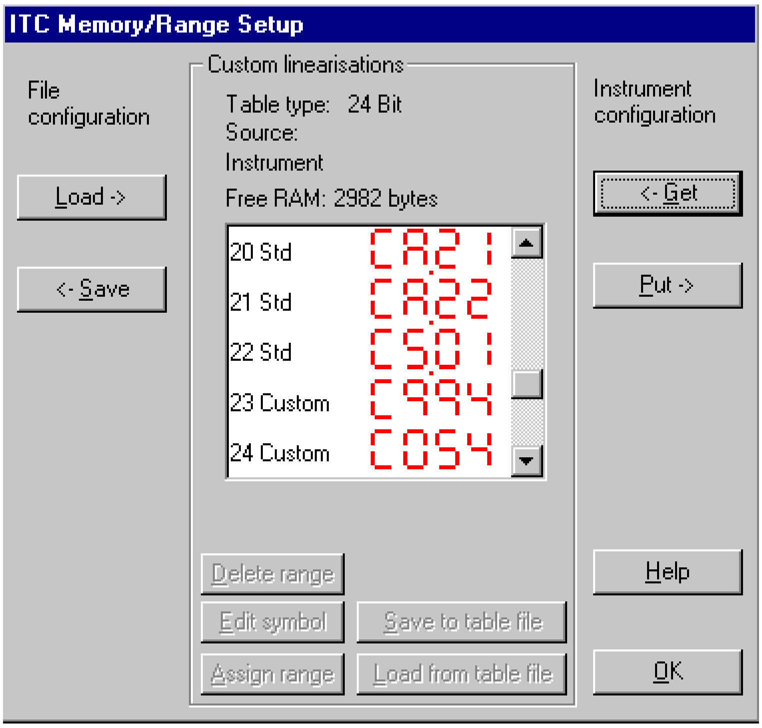


Figure 57 The ITC503 RAM configuration dialog.

The instrument RAM setup dialog is shown in Figure 57. This dialog allows the complete instrument RAM to be read and written, and also loaded and saved to disk files. This feature allows one or more complete ITC503 configurations to be saved and restored. In addition, it is possible to manipulate the instrument's custom linearisations while a RAM map is loaded in the dialog and assign a range to a channel. The example in Figure 57 shows some standard and custom ranges (revealed after use of the "<- Get" button). There are also some unused ranges. The list may be scrolled to see more ranges: it is possible to have up to 32 ranges in total in the ITC503. Custom ranges can also be saved and loaded into the dialog's current RAM.

To read the instrument's RAM, click on the "<- Get" button on the right of the dialog. To read from a RAM file, click on the "Load ->" button on the left of the dialog. In either case, when RAM has been loaded into the dialog, the central box in the dialog will contain information about its custom linearisations. The table type (16 or 24 bits) and the RAM map source (instrument or file) are also displayed. To transfer the RAM map to the instrument or to a disk file, click on the "Put ->" or "<- Save" buttons respectively. In this way, RAM maps may be transferred between disk files and the instrument in a flexible way.

When a RAM map is present in the dialog, it is possible to edit its custom linearisation tables in a number of ways. To operate on a particular table, select it using the mouse - at this point, available buttons will be enabled. Ranges may be written to or read from disk files by selecting the required table with the mouse and clicking on "Save table file" or "Load table file" respectively. You may load a new range into either a "custom" or an "unused" entry - the standard ("Std") ranges may not be overwritten with new ranges. At the top of the dialog box, the free RAM in the instrument is shown. This figure decreases as custom ranges are loaded into the instrument, until it is not possible to load a new range. Typically, there is room for about 8 custom ranges, but this depends on the size of the range data used. Range files may be designed using the "Setup"/"Create Range" menu option described in section 0. You may edit the symbol of the selected range by pressing the "Edit symbol" button, and you may select which sensor channels use the selected range by pressing the "Assign range" button.

When you have downloaded new RAM contents to the instrument, the software will prompt you to transfer the RAM contents to EEPROM for permanent storage when the instrument is switched off.

The ITC503 Range Design Dialog

The ITC503 range design dialog is accessed by selecting the instrument menu option "Setup"/"Create Range", displaying the dialog box shown in Figure 58.

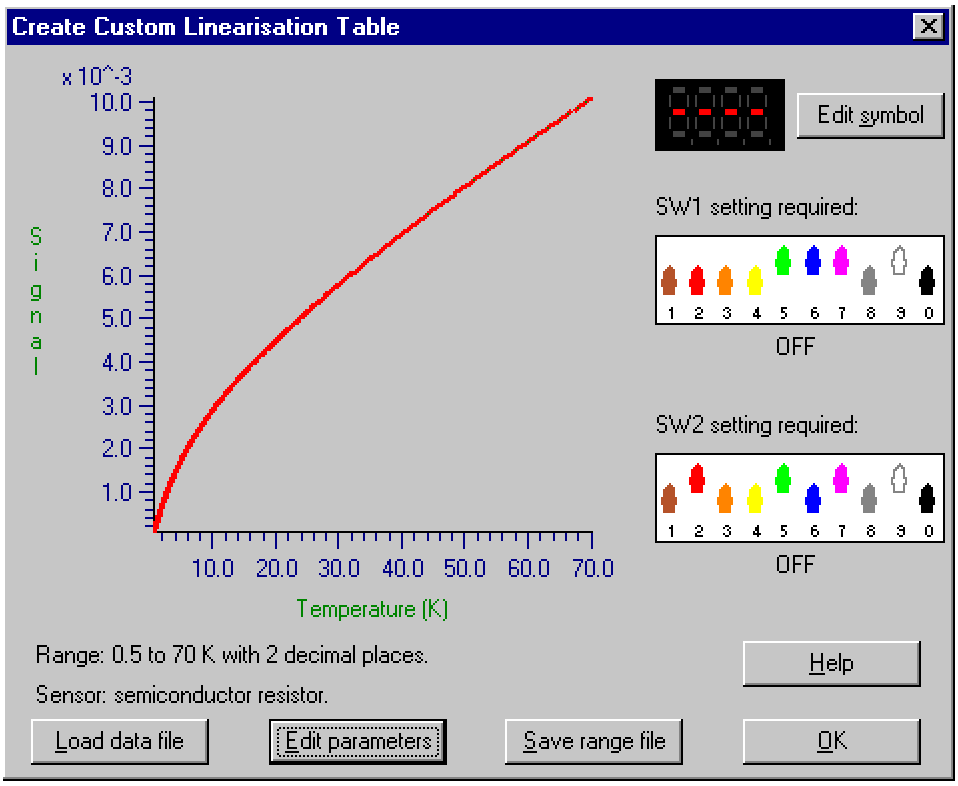


Figure 58 The ITC503 custom range design dialog.

This dialog may be used to convert raw sensor calibration data into the format required by the instrument ready to be downloaded into the instrument RAM set-up dialog (see above). The data should be presented as a standard MMS format data file (see section 0) with a column for temperature data and a column for sensor data (resistance, voltage etc.). All the data files for the standard ranges are provided in a subdirectory named \mmsdata\sys\sensors.

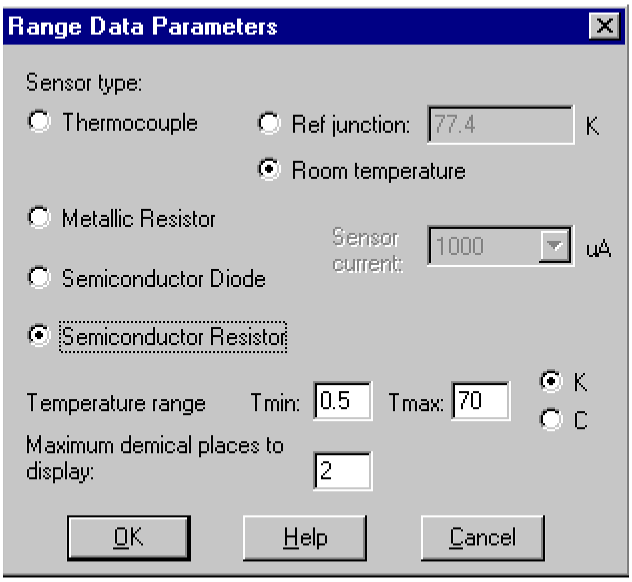


Figure 59 The sensor data dialog box.

First, you must load the sensor data file using the "Load data file" button. If the file is in the accepted format, a further dialog will be displayed asking for further information about the sensor to be used. This dialog is shown in \sensors in Figure 59.
Select the sensor type by clicking on the radio button labelled "Thermocouple", "Metallic Resistor", "Semiconductor Diode" or "Semiconductor Resistor" (See the ITC503 manual for further information on sensor types). Then, enter further data about the sensor in the remaining fields, which are not disabled. For example, \sensors in Figure 1shows that a Semiconductor Resistor is to be used.

The fields at the bottom of the dialog allow the temperature range and maximum number of decimal places displayed to be selected. When you are happy with your selection, click on "OK". You may return to this dialog at any time to change the settings by clicking on the "Edit parameters" button.

At this point, a graph of a function of your raw data (shown as red crosses) will be displayed against temperature. The function of the data used depends on the sensor type, and is intended to make the curve more nearly linear with a positive gradient. Also on this graph will appear a fit to the raw data, shown as a blue line. This is the data that will actually be downloaded to the instrument. To save the custom range data, click on the "Save Range File" button.

When a custom range has been created, the input card DIP switches in the instrument should be set up to correspond to the settings shown on the right of the dialog.

You may edit the symbol for the range by pressing the "Edit symbol" button.
Note that it may not always be possible to find suitable switch settings for the sensor and range that you have requested. This occurs when the sensor input gain or offset required in electronics exceeds that possible. It is then necessary to choose a different temperature range.

IPS120-10 Magnet Power Supply

The IPS120-10 is a superconducting magnet power supply capable of delivering up 120 amps at up to 10 volts. It is intrinsically a bipolar, so that positive and negative fields may be created, and is capable of operating a superconducting switch heater to put the magnet into persistent mode. Its MMS instrument driver allows full remote control, including control of output field or current and ramping rates. Full details of the instrument are included in a separate manual. The name of the MMS driver is "ips120.drv".

Preparing to Use the IPS120-10 from MMS

Figure 60 shows the IPS120-10 instrument driver main window which is displayed by double clicking on the IPS120-10 icon.

Before the software front panel can acquire data from the hardware instrument, you must connect (interface) the instrument to the PC and configure the instrument driver correctly as described in section 0.

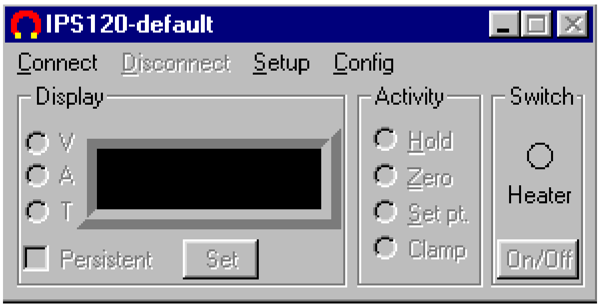


Figure 60 The IPS120-10 software front panel.

Finally, you may "connect" to the IPS120-10 by clicking on the IPS120-10's Connect menu option. The current field or current should appear on the display and be updated in real time. If this is not the case, an error message will be displayed which should give an indication of the nature of problem.

Operating the IPS120-10 from MMS

The IPS120-10 software control panel is shown in Figure 60. It is very similar to the physical instrument front panel and operates in a similar way.
The "Display" box may be used to display field, current or power supply output voltage by clicking on the respective radio button. If the magnet has a superconducting switch fitted and the switch is closed, check the "Persistent" box to display the persistent field or current. The persistent values are based on the current present in the leads last time the superconducting switch was closed.
The "Activity" box contains radio buttons to switch the IPS120-10 into "Hold", "Zero", "Set Point" mode exactly as on the physical front panel of the IPS120-10. "Set Point" mode ramps the magnet to its set point, "Zero" mode ramps the magnet to zero current, and "Hold" mode freezes the IPS120-10 output at its present value. "Clamp" mode indicates that the PSU output is clamped (shorted) which is its power up state, or the result of a magnet quench.

If a superconducting switch is fitted to the magnet and the IPS120-10 has been correctly configured to drive it, the On/Off button in the "Switch" box is available for use. This button toggles the switch heater on and off, and the state of the heater is indicated by the red "LED" above the button. When the heater is on, the switch is resistive, while when the heater is off, the switch is superconducting and the magnet is persistent. You should allow several seconds, depending on the properties of the magnet itself, for changes to the switch heater to take effect.

To enter the required field, click the "Set" button. This reveals a dialog box that allows you to enter the target field or current. If a negative current is set, the polarity changes. The power supply will sweep to the new set field, even if the polarity differs. While one of the IPS120-10's front panel buttons is pressed by a user, the IPS120-10 delays responding to data requests by the PC. If the button continues to be pressed, MMS will time out after 5 seconds.

When you have finished using the IPS120-10 from MMS, click on the Disconnect main menu option to stop communication with the instrument.

The IPS120-10 Setup Dialog

Select the IPS120-10 "Setup" menu option to display the set-up dialog as shown in Figure 61.

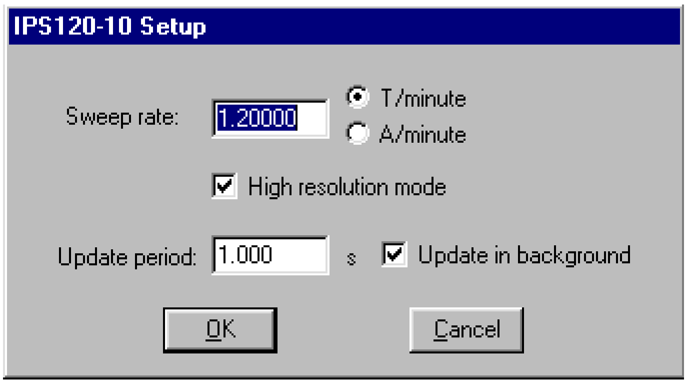


Figure 61 The PS120-10 - dialog.

The set-up dialog allows you to specify the IPS120-10 sweep rate in tesla per minute or amps per minute. The maximum rate that may be used depends on the physical properties of your magnet. If too high a rate is selected, the power supply may voltage limit, or the magnet may quench. If in doubt, contact Oxford Instruments. The setup dialog also allows the update period to be entered. This determines the rate at which new readings are acquired from the instrument and the display updated. A typical value is 2 seconds.

A feature of the IPS120 is its high resolution mode. When this feature is enabled by selecting the "High resolution mode" check box, an additional decimal place for field and current are displayed.

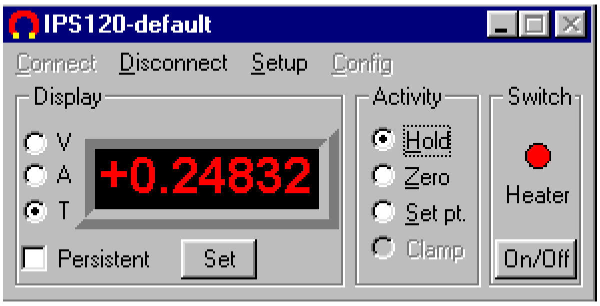


Figure 62 IPS120-10 showing high resolution mode

ILM Intelligent Level Meter

The Intelligent Level Meter model 200 (ILM200) is a family of helium and nitrogen level meters capable of monitoring up to three levels simultaneously. Its MMS instrument driver allows cryogen levels and instrument status to be monitored. Full details of the instrument are included in a separate manual. The name of the MMS driver is "ilm.drv".

Preparing to Use the ILM from MMS

Figure 63 shows the ILM instrument driver main window, which is displayed by double clicking on the ILM icon. Note that the appearance of the main window varies according to the channel configuration of the instrument.

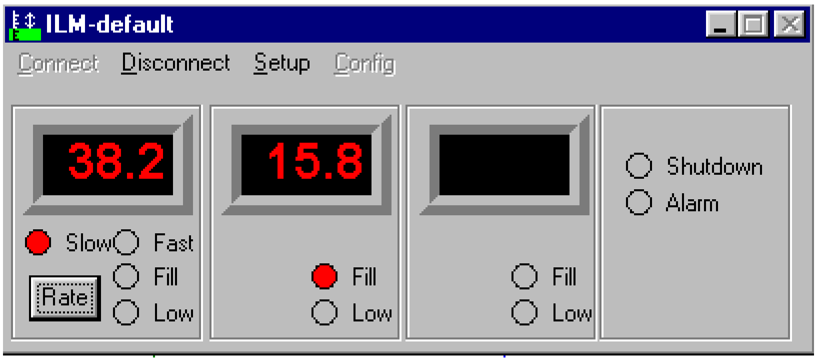


Figure 63 The ILM software front panel.

Before the software front panel can acquire data from the hardware instrument, you must connect (interface) the instrument to the PC and configure the instrument driver correctly as described in section 0.

Finally, you may "connect" to the ILM by clicking on the ILM's "Connect" menu option. The current level should appear on the display and be updated in real time. If this is not the case, an error message will be displayed which should give an indication of the nature of problem. The most likely cause of difficulty is in the interfacing options - if in doubt, review these first.

Operating the ILM from MMS

The ILM software control panel is shown in Figure 63. It is very similar to the physical instrument front panel and operates in a similar way. There are up to three level displays, which configure themselves automatically to correspond to the instrument configuration. Beneath each readout are two lamps, labelled "Fill" and "Low", which indicate that auto fill (see ILM manual) is in progress and that the channel's level is low.

Beneath each Helium channel are two extra lamps and a button, which can be used to control the channel's reading rate as on the actual instrument front panel. Note that under some circumstances, such as when in auto fill mode, only fast update is permitted.

In the status area, to the right of the dialog, are two lamps labelled "Rundown" and "Alarm". These correspond to the same lamps on the actual instrument, which indicate respectively that the magnet is being run down and that the instrument alarm is being sounded.

While one of the ILM's front panel buttons is pressed by a user, the ILM delays responding to data requests by the PC. If the button continues being pressed, MMS will time out after 5 seconds.

When you have finished using the ILM from MMS, click on the Disconnect main menu option to stop communication with the instrument.

The ILM Setup Dialog
The Setup dialog is accessed by selecting the instrument's "Setup" menu option. It allows the sampling rate to be set. The sampling period is the rate at which new readings are obtained from the instrument and the display updated. A typical value is 2 seconds.

Heliox Virtual Instrument
The Heliox virtual instrument is a virtual instrument driver that provides turnkey control of an Oxford Instruments Heliox ^{3}He insert. Automation of ^{3}He charge condensation allows effectively continuous operation of what is normally a single shot cryostat. As well as front panel buttons, macro language commands allows Heliox control to be incorporated very easily into your own programs.

Connecting the Heliox Virtual Instrument
A Heliox can be controlled using one, two or even more ITC’s. There are two common configurations however:

- A single ITC with two sensors (for high and low temperatures) on the ^{3}He pot, a single sensor on the ^{3}He sorb and the heater output can be switched between ^{3}He sorb and ^{3}He pot.
- Two ITC’s: The first ITC monitors ^{3}He pot sensors and has a heater output which can be switched between ^{3}He pot and ^{3}He sorb. The second ITC monitors and controls only the sorb temperature and heater.

Before it can be used, the Heliox virtual instrument must be connected to various signals from the ITC drivers. Additional output signals from the Heliox can then optionally be used in your own application. An example set of signal links is shown below for the two-ITC system described above.

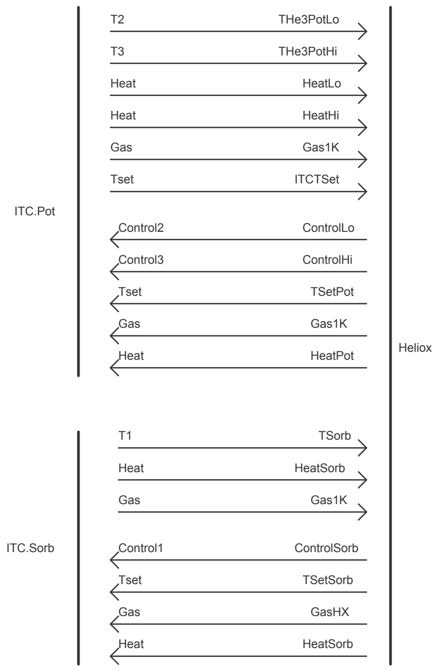


Figure 64 Heliox signal links

In the case of a single ITC system, not all of these links will be used. Simply connect those that are available. The minimum requirements for the Heliox virtual instrument to be operational are:

- Control over sorb heating (output signal heliox.xxx.ControlSorb must be connected)
- A sorb sensor (input signal heliox.xxx.TSorb must be connected)

Automated valves are optional, but will improve the performance of the Heliox. The ability to adjust the 1K pot needle valve allows more complete condensation of the ^{3}He charge. Control over the heat exchanger gas flow will speed up the condensation process, and provide more flexibility in the range of operating temperatures. High temperature control is difficult with only a single ITC since the sorb temperature, and hence the thermal link between 1K pot and ^{3}He pot cannot be controlled.

- At least one sensor on the ^{3}He pot (both inputs heliox.xxx.THe3PotHi and heliox.xxx.THe3PotLo must be connected, but this can be to the same ITC output)

Using the Heliox Virtual Instrument

Having connected the Heliox virtual instrument as described above, you can use it to automate the operation of the cryostat. Figure 65 shows the main instrument dialog. On the right of the dialog, a schematic diagram of a Heliox is used to display temperatures and gas flows. Starting from the top, these are the sorb temperature, the 1K needle valve setting, the 1K pot temperature, the heat exchange gas flow and the ^{3}He pot temperature. Below the ^{3}He pot temperature is the stability status. This can be either “Not set” (when the Heliox does not have a set point), “Stable” or “Stabilising”. Stability is explained further in section 0. On the left of the dialog are the Control panel and two other panels showing further status information.

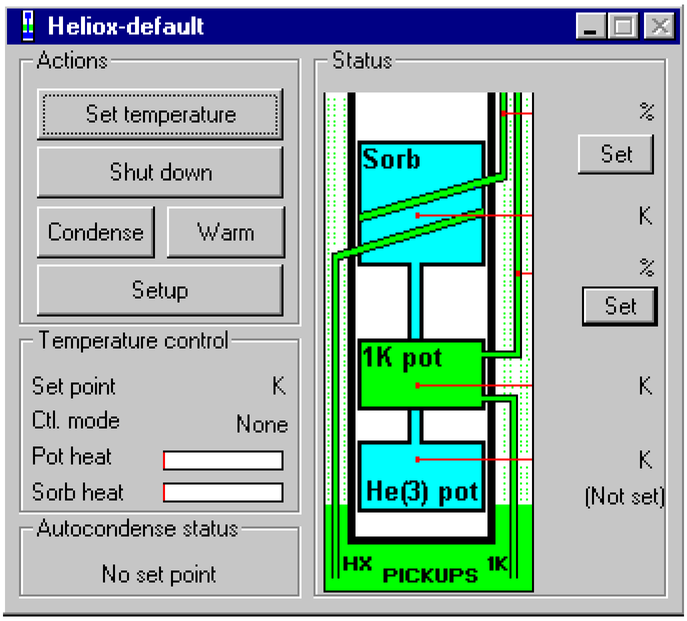


Figure 65 Heliox virtual instrument

Control Panel

The buttons in the Control panel provide control over the Heliox. The Set Temperature button allows you to enter a new set point. The Setup parameters, described below, are used to decide how control is to be achieved at this temperature. Setting a low temperature may also trigger a recondensation of the ^{3}He as described in section 0. If the temperature entered is greater than the Maximum Heliox Temperature specified in the Setup, a dialog will alert you of the fact, and no further action will be taken.

The Shut down button allows you immediately to turn off all heaters connected with the Heliox. This can be used if incorrect set-up parameters have caused an excessive amount of heat to be applied to the Heliox. All heaters will be set to Manual control and zero output. To regain temperature control, you need to enter a Set Temperature again.

The Condense button unconditionally starts the ^{3}He recondensation procedure described below. The Warm button pumps out the 1K pot as a precursor to warming the Heliox to room temperature. The Setup button allows you to edit the Heliox set-up information described in Heliox Setup dialog.

Temperature Control Panel

The Temperature Control panel shows how the Heliox temperature is being controlled. The Set Point is simply the temperature at which the Heliox virtual instrument has been asked to control. The Ctl. Chan. field shows the control channel being used to obtain this temperature. It can take on the values “Pot Low” or “Pot High” and “Sorb". “Pot Low“ uses sorb heating to control the pot temperature. “Pot High” uses a heater on the ^{3}He pot directly to control its temperature. ”Sorb” indicates that the sorb temperature is being directly controlled at a particular value. The Sample Heat and Sorb Heat fields simply show bar graphs of the relevant heater output, on a scale of 0 to 100 per cent.

The information on this panel relies on the Heliox virtual instrument knowing the true state of the ITC’s used to control the Heliox itself. If you have used the Set Temperature button (or the Heliox.Tset signal) this will automatically be the case, provided your Signal Links have been correctly set up. When you first connect the ITC’s used by the virtual instrument however, the Heliox obtains the set point from the ITC via the ITCTSet signal. It is then assumed that the ITC’s are configured as they would have been if this temperature had been entered using the Set Temperature button, and displays are updated accordingly. For example, if the set point from the ITC is 0.5K and this is below the Use He(3) Below temperature in the Setup, then the Heliox virtual instrument assumes that the sorb is being heated using feedback from the ^{3}He pot. The sorb heat display is therefore updated using values obtained from the HeatLo signal. If heat is being supplied to the sorb from a separate ITC this is not shown on the display.

If the Heliox has no set point then the sorb heater display is updated from both possible inputs (HeatLo and HeatSorb). If you have two ITC’s and they have different heater outputs then the Sorb Heat display will fluctuate between these two values, since either could be supplying heat to the sorb. The Pot Heat will always show the value linked to HeatHi, which is usually the same as the HeatLo value.

Autocondense Status

The Autocondense Status can display one of five messages depending on the configuration and status of the Heliox. Note that these conditions are checked for in the order shown, and subsequent conditions are not checked. If there is no set point for example, the message “No set point” is displayed and it is not possible to tell from the main dialog whether or not autocondensation has been enabled. The possible messages are given below:

- “No set point”
  The Heliox does not have a temperature set point and will therefore not perform an autocondense.

- “Set pt outside He(3) range”
  The set temperature is higher than the ^{3}He threshold value defined in the setup – autocondense will not be performed.
- “Condensing”
  The Heliox is condensing in the ^{3}He charge. The Condense dialog should also be visible.
- “Disabled”
  Autocondensation has been disabled from either the Setup dialog or the Autocondense signal.
- "Dead time: XXXs"

Condensation has recently been performed and has been temporarily disabled for the duration specified by the Deadtime parameter in the Condense Conditions section of the Setup dialog. The time display counts down to zero at which point autocondensation is re-enabled.

Heliox Setup dialog

The Heliox set-up dialog allows you to specify the configuration of your Heliox and to control the way in which it operates. There are five sections to the set-up – Needle Valves, Operational Parameters, Stability Monitoring, Condense Conditions and Condense Parameters.

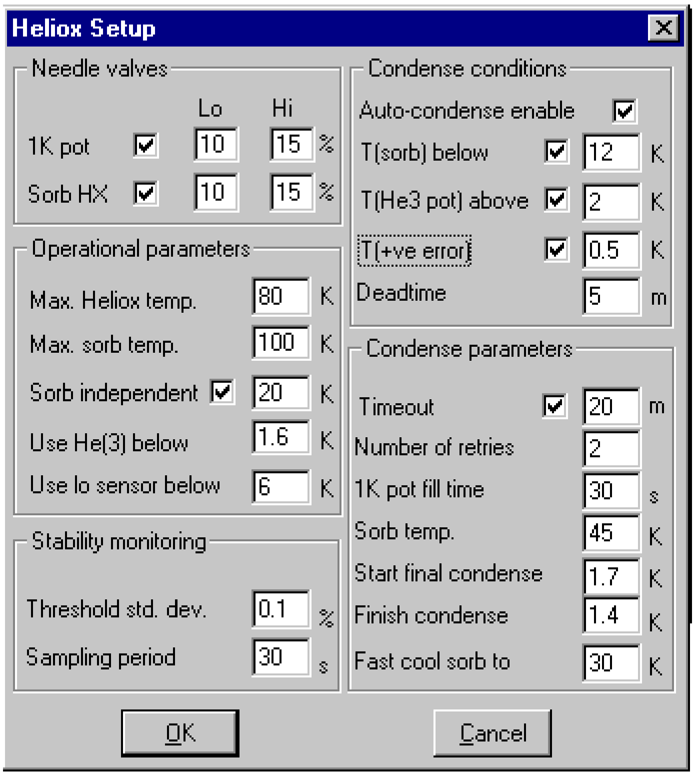


Figure 66 Heliox setup dialog

Needle Valves

The Needle Valves section allows you to specify whether or not you have automated needle valves for the two gas flow control points on a Heliox, and what values to use for them when setting a temperature.

On the older style Heliox inserts the 1K pot needle valve is used to control the rate at which ^{4}He is sucked from the main bath into the 1K pot. This is an important parameter. If the rate is too low then the pot will eventually run dry and temperature control will rapidly be lost. If it is too high then performance will be decreased by the excessive inflow of relatively warm ^{4}He, and helium consumption will also be increased.

The new style Heliox gives better temperature stability by replacing the 1K pot with a plate and a continuous flow ^{4}He coil heat exchanger. These inserts are controlled using only one ITC503 and a heater control box. In the dialog in Figure 66, uncheck the 1K pot needle valve to invoke the control algorithm for the new style heliox. The signal link configuration changes as there is no 1K pot needle valve or 1K pot sensor. The Sorb HX valve controls the flow of helium gas past the ^{3}He sorb heat exchanger. This determines the cooling power on the sorb, and consequently the amount of heat required to maintain a given sorb temperature. There is a secondary effect in that too low a flow will create a large heat load on the 1K plate coil due to conduction of heat down the slow-moving column of gas.

Both valves have a check box that tells the software whether the valve is present. There are also two settings for each valve that are used when the Heliox is being operated at low and high temperatures respectively. Low temperatures are defined as those lower than specified in the Use He(3) Below parameter. These values will be greyed out if the associated check box is deselected. Having a 1K needle valve affects the way in which condensation proceeds (this is discussed in more detail in section 0). If there is no 1K needle valve, then the 1K Pot Fill Time and Finish Condense parameters will be greyed out since they are no longer relevant. Similarly, deselecting the Sorb HX valve will grey-out the Fast Cool Sorb To parameter.

Operational Parameters

These are general operating conditions for the Heliox.

The Max. Heliox Temp. and Max. Sorb Temp. parameters are provided as a safety measure. If either is exceeded at any time, the Heliox is put into manual control and all heat is switched off. The sorb temperature limit is particularly useful because of the possibility of thermal runaway, described in the Heliox manual. If the sorb heater is being used in an attempt to control the ^{3}He pot temperature at too high a value, then the sorb temperature can rise without limit while the pot temperature remains below its set point. This could ultimately cause damage to your system.

The Use He(3) Below value is used to specify a threshold temperature below which ^{3}He cooling is used, and above which ^{4}He cooling is used. In the former case, the ^{3}He sorb heater is used to control the ^{3}He pot temperature. In the latter case a heater on the pot is used directly to control its temperature.

Use Lo Sensor Below applies to systems with two sensors on the ^{3}He pot. Since there will generally be a region of overlap between the sensors, this specifies the temperature at which to make the changeover. Note that the low temperature sensor will generally saturate at some value and you should make sure that the threshold value you specify is below this since otherwise the high temperature sensor may never be selected.

Stability Monitoring

Stability monitoring allows a stability flag to be set if the standard deviation of the ^{3}He pot temperature falls below a specified limit.

The Threshold Std. Dev. parameter is used as the comparison value in deciding whether the temperature is stable. Not only the standard deviation, but also the difference between the ^{3}He pot temperature and the set point, must be equal to or lower than this value. Note that it is specified as a percentage of the pot temperature since, for example, 1mK may be a realistic limit at 0.5K but not at 50K.

The Sampling Period specifies the period in seconds over which the standard deviation is calculated. The number of points received in this time will depend on the update rate of the ITC in question. Because of performance considerations, the maximum number of points used is limited to 200. If more points than this are collected in the specified period, an error message will alert you to the fact. You should then reduce either the Sampling Period or the update rate of the ITC.

Condense Conditions

The Condense Conditions section allows you to define under what circumstances an auto-condense will be performed. There are three conditions, each of which can be enabled or disabled with a checkbox as well as being varied with a numerical value. Autocondensation can be completely disabled by deselecting the Auto-condense Enable checkbox.

During operation at ^{3}He temperatures, the Heliox's charge of ^{3}He is gradually boiled off from the pot and absorbed into the charcoal sorption pump. In order to maintain suction as it becomes saturated, the sorb temperature will lower steadily if temperature control of the sorb is being used to keep a constant pot temperature. At about 10K, maximum pumping is achieved and if the sorb temperature approaches this value it is a good indication that the ^{3}He charge is almost finished. This condition is checked with the T (sorb) Below parameter.

When the liquid ^{3}He is finished, the pot temperature will no longer be maintained, and will rise above the set point. This condition can be detected with the T(pot) Above parameter (which looks for the temperature exceeding a given absolute value) and T(+ve error) which looks for the pot temperature exceeding the set point by more than the specified value.

The Deadtime parameter allows a period of recovery after an autocondense has been performed. In re-establishing temperature control after condensing in the ^{3}He, the sorb temperature will often fall below 10K, and the ^{3}He pot temperature will oscillate about the set point. Since this could otherwise trigger an autocondense, the Deadtime specifies a period after condensation during which autocondensation is temporarily disabled.

Condense Parameters

The Condense Parameters section specifies the way in which condensation is performed. This is independent of how the condensation was triggered – whether by one of the conditions described above, or by a button press. Please see section 0 for a description of the actual recondensation process.

The Timeout parameter is used to detect a failure to recondense. If this option is selected, and condensation has not been successfully completed within the specified time, then the process is aborted. Note that recondensation will not timeout during the final stage of cooling the sorb since the actual condensation of ^{3}He has already finished at this stage.

On systems with an automated 1K needle valve, the condensation process is started by opening completely the needle valve in order to start with a full 1K pot. The 1K Pot Fill Time defines how long this is done for.

The Sorb Temp. specifies a sorb temperature to be maintained during condensation. The value typically used is about 45K. It should be high enough to ensure that nearly all the ^{3}He is driven out of the sorb. Note that during condensation some leeway is allowed on this temperature.

The Start Final Condense value defines the pot temperature at which the 1K needle valve (if present and if automated) is closed down prior to the final condense. If the needle valve is not present a different algorithm is used. If it is present but not automated, then this is the end of condensation, otherwise Finish Condense defines a further pot temperature at which condensation is deemed to have finished.

Finally, on systems with an automated heat exchanger needle valve, the valve is opened completely until the sorb temperature has dropped below the Fast Cool Sorb To parameter. This is purely a timesaving measure which allows the sorb to reach its normal control temperature more quickly than would otherwise be the case.

^{3}He Recondensation

Recondensation is the process of liquefying the ^{3}He charge of the Heliox prior to using it for cooling the cryostat below 1K or so. This section describes the process as implemented in the software.

Efficient recondensation involves achieving the lowest possible ^{3}He pot temperature combined with the highest possible ^{3}He sorb temperature. When this has been achieved, the maximum possible fraction of the Heliox's ^{3}He charge will exist as liquid in the ^{3}He pot. This will give the longest possible hold time for the ^{3}He pot. In practice, it is found that there is little to be gained from sorb temperatures in excess of 40K or so. The objective therefore is simply to obtain the lowest possible pot temperature while controlling the sorb at about 40 - 45K. Since different systems will perform differently, the obtainable pot temperatures will vary. In determining which values to use for your system, you must strike a balance between achieving both long hold times and quick, reliable recondensation.

Figure 67 shows the dialog that appears during recondensation. The graph plots pot and sorb temperatures on different scales, and these temperatures are also displayed numerically in panels below. The current activity is shown in the Status panel, and a Cancel button allows the process to be aborted.

Before recondensation begins, you are given a few seconds to cancel out of the dialog before any action is taken. This feature is provided to save you having to wait for the temperature to restabilise if you have accidentally triggered condensation, either by pressing the Condense button or through incorrect setup parameters.

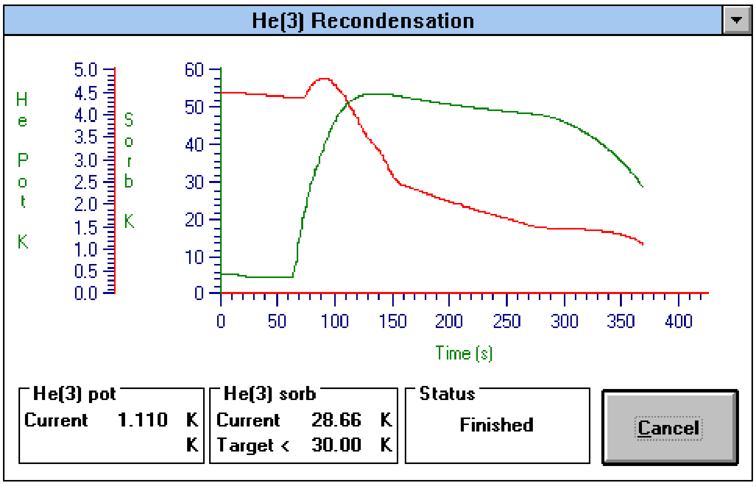


Figure 67 ^{3}He recondensation

If there is an automated 1K pot needle valve, the first step is to open it fully for the time specified by the 1K Pot Fill Time Parameter to allow the pot to fill with helium liquid. The needle valve is then set to its low temperature value. The sorb is set to control at the temperature specified in the set-up – usually about 45K – while waiting for the pot temperature to fall below the Start Final Condense value. Once this has been achieved, and the sorb is close to its set point, the first stage of condensation has been completed. If there is an automated 1K pot needle valve, this is then closed completely. The new target then becomes the Finish Condense value. In the absence of an automated 1K pot needle valve however, sorb cooling is performed immediately after the Start Final Condense value is reached.

Sorb cooling is the final stage of the condense procedure. It consists simply of opening the heat exchange valve completely until the sorb temperature falls below the value specified by the Fast Cool Sorb To parameter. This stage saves a significant amount of time compared with waiting for the sorb to cool with normal gas flow, and allows your experiment to continue as soon as possible. It can only be performed with an automated heat exchange valve. If there is no such valve, this step is skipped.

After condensation has finished, temperature control is resumed at the current set point, and the valves are returned to values appropriate to that temperature as defined in the set-up. If you Cancel out of the condensation dialog, the heaters are turned off, and the needle valves set to their low temperature values. The main dialog will show "No set point" and further recondensations will be disabled until you reset the temperature. If the condensation times out, a message will appear in the condensation dialog, and the dialog will remain until you press Cancel.

The key points illustrated by the macro are:

- The Heliox temperature is set with the simple command **heliox.tset = t**, independently of how this temperature is achieved (whether by ^{3}He or ^{4}He cooling). Achieving this temperature may involve condensing the ^{3}He charge.

- A simple two line loop delays measurements until the set temperature is reached and the temperature has stabilised. Achieving the set temperature may involve a change in control mode, or even a recondensation, but the macro language does not have to know about these things – the only effect is to increase the time taken.
- Recondensation is disabled during measurements to prevent an autocondense and then re-enabled afterwards. Note that if recondensation is immediately triggered upon enabling it, the next set point is stored by the condensation dialog and set on completion.

Appendix 3: Oxford Instruments ISOBUS

Note: This text duplicates information provided with all Oxford Instruments electronics products.

A unique feature of ITC503 and other Oxford Instruments products, is the ability to connect a number of instruments simultaneously, to a single RS232 port on a computer and to control each one independently. This is done by means of an ISOBUS cable which carries a single MASTER connector (25-way D socket) and up to eight, daisy-chained SLAVE connectors (25-way D plugs). Each slave connector incorporates full optical isolation so that the slaves are all isolated from the master and from each other. The slave connectors draw their power from the individual instruments, via the DCD signal on pin 8. The master connector may draw its power from either DTR or RTS signals from the computer.

To use ISOBUS, a special communication protocol is required, which is part of the command structure of Oxford Instruments' products. To allow separate instruments to be distinguished, each is allocated a unique address in the range 0 to 9, held in non-volatile memory. By default, an ITC503 is supplied with the Isobus address of 1. When operating on ISOBUS an instrument must be able to recognise and respond to commands addressed to it, whilst ignoring commands addressed to other instruments. This is achieved by starting all commands with a special ISOBUS control character.

When more than one powered-up instrument is connected on ISOBUS, no command should be issued which does not have an ISOBUS control character as its first character. Issuing such a command would result in an unintelligible response, as all instruments would reply together. (N.B. This will only result in lost data. No hardware damage will be caused).

Following the control character and its parameter (where required), the rest of the command follows the form described above. The response of the instrument depends on the initial control character in the following manner:

@n (At) addresses the command to instrument number n, where n is a digit in the range 0 to 9. This instrument obeys the command and returns its usual response. All other instruments ignore the command and send no reply.

$ (Dollar) instructs all instruments to send no reply. This is normally used to precede a command being sent to all instruments simultaneously, and prevents a conflict as they all echo the command together.

It may also be used in non-ISOBUS applications if the computer does not wish to receive a response. It should be used with caution however, since all responses are suppressed, including the "?" error response. Thus, the computer has no way of knowing if a command has been received or even if the instrument is connected.

If a command is to be addressed to a specific instrument, but no reply is required, it is permissible to use "$" and "@n" together. The "$" should always come first.

& (Ampersand) instructs an instrument to ignore any following ISOBUS control characters. It is included in the ISOBUS protocol to allow instruments whose command repertoire includes "@", "$", "&" or "!" to be used on ISOBUS. ITC503 does not require the use of this command.

!n (Exclamation) instructs the instrument that from now on its address is to be n. This command is included here since it is relevant to ISOBUS operation. However for obvious reasons, it should not be sent when more than one instrument is powered up and connected to ISOBUS. (It would result in all instruments having the same address!). The command is intended for initial setting up of instruments, one at a time. To avoid inadvertently changing addresses, the "!" command will only be obeyed following a "U" command with a non-zero password. Note that the address set this way is the ISOBUS address, not the GPIB address (if applicable).

Appendix 4: Troubleshooting

Sequence fails to run and gives the message "Out of memory, error no. 7"
For long and complex sequences you may have to increase the "Free memory" available when executing BASIC. The default is 260 Kbytes but this can be increased as follows:

Use a text editor to open the file "oiri.ini" (which will usually be found in the Windows directory). In the section headed by
[ObjectBench] (note square brackets)
Insert the line
BasicFreeMemory=1000000
where the number indicates the new value for Free memory.
You should save the edited "oiri.ini" and restart MMS.