



a Roxboro Group company

SMaRT

USER GUIDE

Part No.

Issue: AA

Date: Feb. 2003

Solartron Analytical UK

Unit B1 Armstrong Mall,
Southwood Business Park,
Farnborough,
Hampshire,
GU14 0NR
Tel: +44 (0) 1252 556 800
Fax: +44 (0) 1252 556 899
E-mail info@solartronanalytical.com
E-mail for service issues
service@solartronanalytical.com
E-mail for customer support issues
support@solartronanalytical.com

Solartron US

19408 Park Row, Suite 320
Houston, Texas, 77084
Tel: (1) 281-398-7890
Fax: (1) 281-398-7891

Web: <http://www.solartronanalytical.com>

For details of our agents in other countries, please contact our Farnborough, UK, office.

*Solartron pursues a policy of continuous development and product improvement.
The specification in this document may therefore be changed without notice.*

Contents

1.	SMART CONCEPTS.....	1-1
1.1	THE SMART USER INTERFACE	1-1
1.1.1	SET-UP ENTRIES	1-1
1.1.2	SMART STRUCTURE	1-2
1.2	SMART PROCESSING AND DISPLAY FACILITIES	1-3
1.2.1	SELECTING THE PLOT TYPE	1-3
1.2.2	SETTING UP THE TRACES	1-3
1.2.3	SETTING UP THE X-AXIS	1-4
1.2.4	SETTING UP THE Y-AXES	1-4
1.2.5	SETTING UP THE VIEW	1-4
2.	USING SMART	2-1
2.1	CREATING YOUR PROJECT	2-3
2.2	SPECIFYING THE SYSTEM HARDWARE.....	2-4
2.3	DESIGNING YOUR TEST SEQUENCE	2-5
2.4	STARTING AN EXPERIMENT	2-6
2.5	VIEWING THE RESULTS OF YOUR EXPERIMENT	2-7
3.	SPECIFYING THE SYSTEM HARDWARE.....	3-1
3.1	ENTERING THE HARDWARE SET-UPS.....	3-1
3.1.1	SETTING THE DIELECTRIC INTERFACE	3-2
3.1.2	SETTING THE FREQUENCY RESPONSE ANALYSER (FRA)	3-2
3.1.3	SETTING THE TEMPERATURE CONTROLLER	3-3
4.	CREATING AND MANAGING A PROJECT	4-1
4.1	PROJECT FILES.....	4-1
4.2	CREATING A PROJECT	4-2
4.3	MANAGING A PROJECT	4-4
4.3.1	LOADING A PROJECT.....	4-4

4.3.2	ADDING FILES FROM OTHER PROJECTS	4-5
4.3.3	SAVING A PROJECT	4-5
5.	CREATING AN EXPERIMENT.....	5-1
5.1	USING THE EXPERIMENT EDITOR	5-1
5.2	CREATING AN EXPERIMENT	5-3
5.3	DEFINING THE SAMPLE.....	5-6
5.4	SETTING UP A MEASUREMENT	5-7
5.4.1	SELECTING THE ANALYSER INTEGRATION (1294/1296)	5-8
5.4.2	SELECTING THE MEASUREMENT METHOD (1296)	5-9
5.4.3	ANALYSER INPUT RANGING AND COUPLING (1294)	5-10
5.4.4	GENERATOR BALANCING (1294)	5-11
6.	RUNNING AN EXPERIMENT	6-1
6.1	STARTING AN EXPERIMENT	6-1
6.2	STOPPING AN EXPERIMENT	6-2
7.	VIEWING YOUR RESULTS.....	7-1
7.1	PREPARING TO VIEW	7-1
7.2	VIEWING YOUR RESULTS ON A GRAPH.....	7-2
7.2.1	SELECTING THE PLOT TYPE	7-2
7.2.2	SETTING UP THE TRACES	7-3
7.2.3	SETTING UP THE X AXIS	7-4
7.2.4	SETTING UP THE Y AXES.....	7-5
7.2.5	SETTING UP THE VIEW.....	7-6
7.2.6	ZOOMING IN ON THE GRAPH DETAIL	7-8
7.3	TABULATING YOUR RESULTS	7-9
7.4	CONVERTING THE DATA FILE FORMAT	7-10
7.5	NORMALISING THE MEASUREMENT DATA	7-11
7.5.1	NORMALISATION PROCEDURE	7-12
8.	SUPPORT FEATURES	8-1
8.1	USING THE SUPPORT FEATURES.....	8-1

8.2	SELECTING THE GENERAL OPTIONS	8-2
8.3	SELECTING THE VIEW OPTIONS	8-3
8.4	SELECTING THE MESSAGES OPTIONS	8-4
8.5	SELECTING THE GRAPHICS OPTIONS.....	8-5
8.6	SEARCHING THE GPIB FOR LISTENERS.....	8-5
9.	INDEX	9-1
	<i>Figure 1-1 The 1296 tool bar and menus.</i>	<i>1-1</i>
	<i>Figure 5-1 Generator balance for four-terminal measurement.</i>	<i>5-11</i>
	<i>Figure 7-1 Graph configuration: plot type.....</i>	<i>7-2</i>
	<i>Figure 7-2 Graph configuration: trace set-up.</i>	<i>7-3</i>
	<i>Figure 7-3 Graph configuration: x-axis.....</i>	<i>7-4</i>
	<i>Figure 7-4 Graph configuration: y-axes.....</i>	<i>7-5</i>
	<i>Figure 7-5 Graph configuration: view.</i>	<i>7-6</i>
	<i>Figure 7-6 Example of a Bode plot.....</i>	<i>7-7</i>
	<i>Figure 7-7 Example of graph zooming.</i>	<i>7-8</i>
	<i>Figure 8-1 General options window.</i>	<i>8-2</i>
	<i>Figure 8-2 View option window.</i>	<i>8-3</i>
	<i>Figure 8-3 Project tree styles.</i>	<i>8-3</i>
	<i>Figure 8-4 Messages options window.....</i>	<i>8-4</i>
	<i>Figure 8-5 Graphics option window.....</i>	<i>8-5</i>

1. SMaRT Concepts

1.1 THE SMART USER INTERFACE

Broadly, there are three steps in using SMaRT: setting up the system, making the measurements, and displaying the results. SMaRT clearly relates the system set-up and operation to what you are trying to achieve in your experiment and enables you to concentrate purely on the measurement results. An introduction to the SMaRT structure is given below.

1.1.1 SET-UP ENTRIES

The logically arranged tool bar and menu structure of SMaRT enable you to set up an experiment quickly and easily. Figure 1-1 shows the general arrangement.

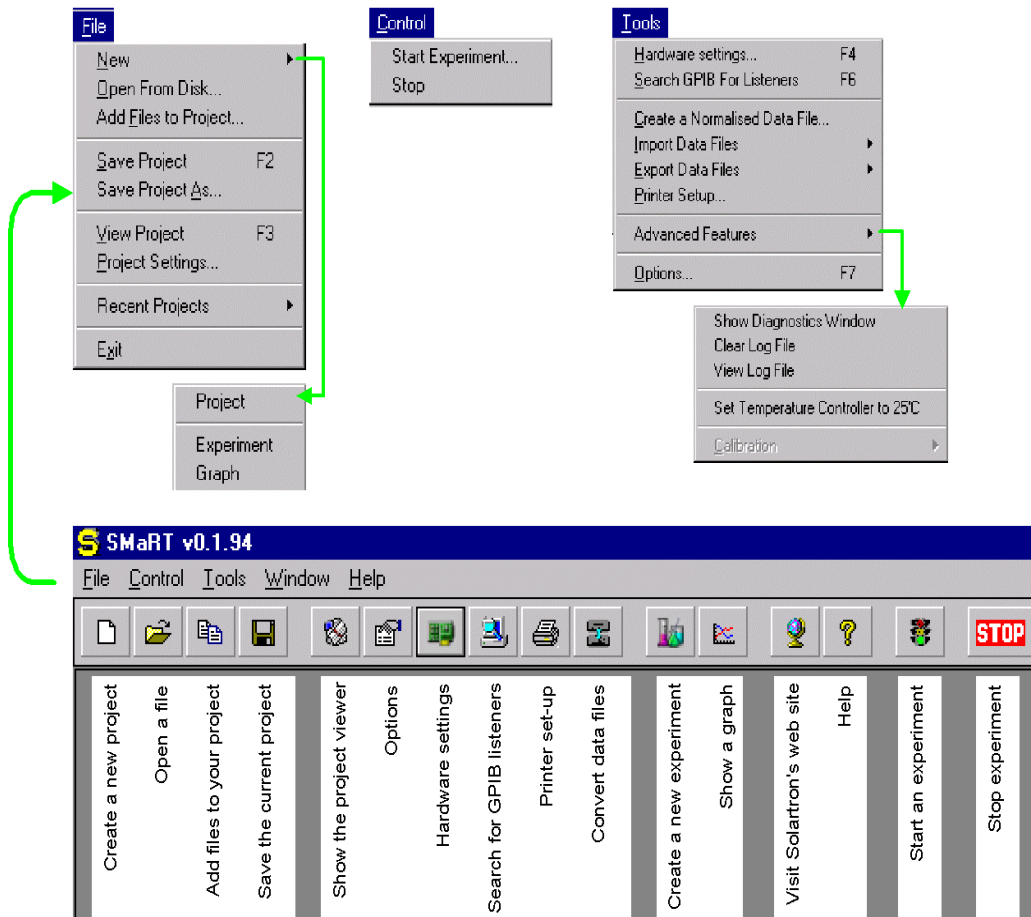


Figure 1-1 The 1296 tool bar and menus.

Many of the tool icon and menu entries lead you into a clearly presented window, or series of them, on which you set up your experiment.

Many of the tool icon and menu entries lead you into a clearly presented window, or series of them, on which you set up your experiment.

1.1.2 SMART STRUCTURE

To simplify the 1296 set-ups, the structure of SMaRT is based on *projects* and *experiments*.

1.1.2.1 The Project

A project comprises a set of files relating to a common aim –for example, ascertaining the properties of a particular type of material. The **Project Viewer** represents the project on the **SMaRT** display. This shows all the files presently included in the project, in the form of icons. The types of files used are:

- Experiment file – specifies the hardware to be used for a particular experiment.
- Data file – contains the data obtained by running an experiment.
- Graph file – specifies a graph configuration and points to the data file containing the data to be plotted.
- Report file – contains a summary of an experiment that has been run.

A project may embrace several experiments and each one will have its own data and report file assigned to it.

1.1.2.2 The Experiment

To create an experiment you use the **Experiment Editor**. Here you define:

- The filename of the experiment file, and where to save it,
- The file to be used for the experiment data,
- The experiment parameters, such as ac level, frequency and temperature

Having created an experiment, you must save it to make it available. An experiment icon then appears on the **Project Viewer**. Any experiment thus created can then be run to acquire the materials testing data. The ways in which you can view this data are introduced in Section 1.2.

A summary of the SMaRT set-up and data acquisition and display procedure appears in Section 2.1 of this user guide.

1.2 SMART PROCESSING AND DISPLAY FACILITIES

When you run an experiment the basic voltage and current data are stored in memory. SMaRT is able to retrieve this data, and to process and plot the data in near real-time or historically. A detailed analysis of cell performance is thus provided, in graphical or tabular form.

You set up the graph you wish to plot on the graph set-up window. The procedure for this is given in Section 7.

1.2.1 SELECTING THE PLOT TYPE

On the *Plot Type* tab, you select the type of graph you want. This can be for *Impedance plots*, *Bode* or *Complex plane*. On the other five tabs you proceed from left to right, setting up the trace(s), x-axis, y-axis, etc, as explained in the following sections. When graph configuration is complete, click on **OK** to display the graph.

1.2.2 SETTING UP THE TRACES

On the *Trace Set-up* tab, you assign a data file to each trace to be plotted on your graph. Up to eight traces may be plotted on the same graph.

For each trace, you follow the same simple set-up procedure:

1. Select a trace on which to plot the data.
2. Attach the trace to the relevant y-axis, **Y1** or **Y2**. (The **Y2** axis may be enabled, or not, for *Impedance* plots.)
3. Select the data file containing the data you wish to plot.
4. Select the range of data you wish to plot. (For time related data, for example, you can define the period that the plot is to cover.)

In step 4 of the trace set-up procedure, you have the option of selecting:

- For historic data:
- The date and time when the data was obtained,
- The number of times you wish to plot a repeated experiment,
- The period of the data to be plotted

1.2.3 SETTING UP THE X-AXIS

The type of units you can plot on the x-axis depends on the selection you make under **Plot Type**. By default, the x-axis is in units of **Frequency** for **Impedance** plots.

You can select alternative units to suit your needs. These units include magnitude, phase and real for impedance plots. Full lists of the units available are shown in Table 1-1.

1.2.4 SETTING UP THE Y-AXES

The type of units you can plot on the y-axis depends on the selection you make under **Plot Type**. The default units for **Impedance plots** are Y_1 =magnitude and Y_2 =phase. You can select alternative units to suit your needs. These units include impedance real and imaginary, or admittance magnitude and phase for impedance plots.

1.2.5 SETTING UP THE VIEW

On the **View** window, you set up the general appearance of your graph.

The **Display** options apply to all graphs. With these, you can:

- Display the grid, or not,
- With the grid displayed, display the minor divisions, or not,
- Display the cross-wire cursors, or not,
- Display the legends (trace labels), or not.

The **Complex plane options** are available only with **Complex plane** selected as the **Plot Type**. These options allow you to:

- Use isotropic scaling (equal scaling on x- and y-axes), or not,
- Invert the y-axis, or not.

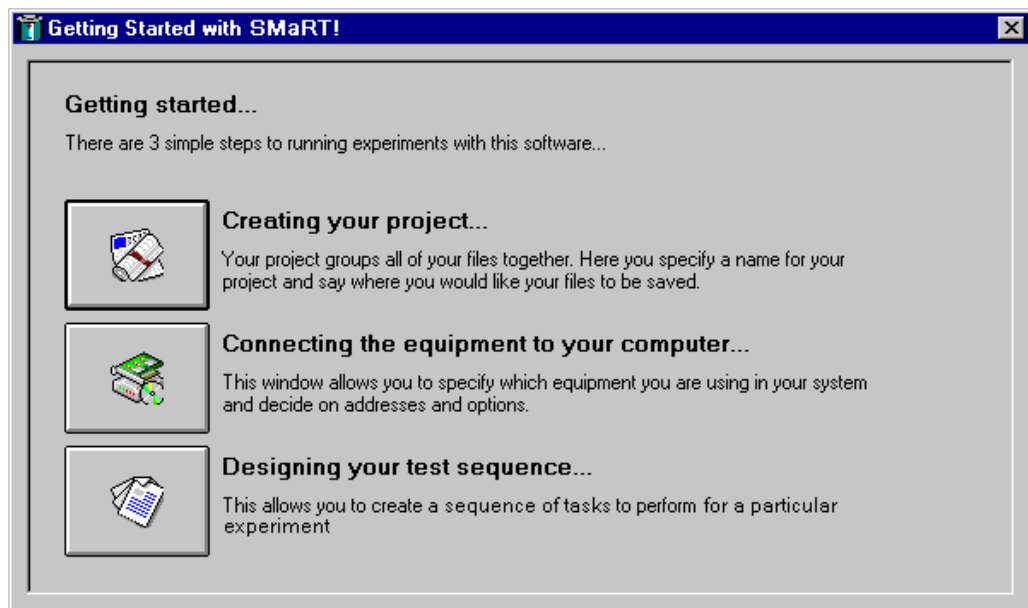
2. Using SMaRT

On starting SMaRT, you will see the *Welcome to Smart* panel:



This gives you three options and the choice you make will depend on whether you are a new or experienced user.

New users should choose *Getting Started!* This presents you with the few simple steps you will follow to create your first project:



The aim of this is to get you started in running tests. We recommend that you first connect a sample of the material under test, as described in the 1296 User Guide. Then use the Getting Started procedure outlined below to get to know the facilities offered by the 1296 Unit and SMaRT.

To create your project, click in turn on each button on the *Getting started...* panel, starting at the top with the *Creating your project*. You thus progress

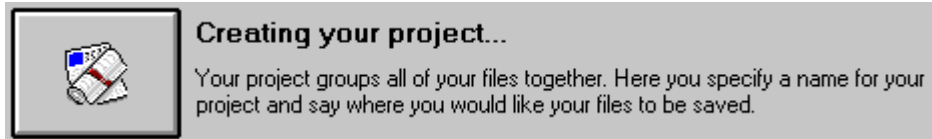
through a simple three-step procedure. Sections 2.1 through 2.3 below introduce you to the type of windows you will encounter in setting up a project and you are shown how you create and manage *projects* and *experiments* within SMaRT. In each section, reference is made to more detail in later sections of the User Guide.

Having set up your initial project, you can start your experiment. Section 2.4 shows you how. On the successful completion of the experiment, you can view the data acquired. Section 7 of the User Guide gives full details.

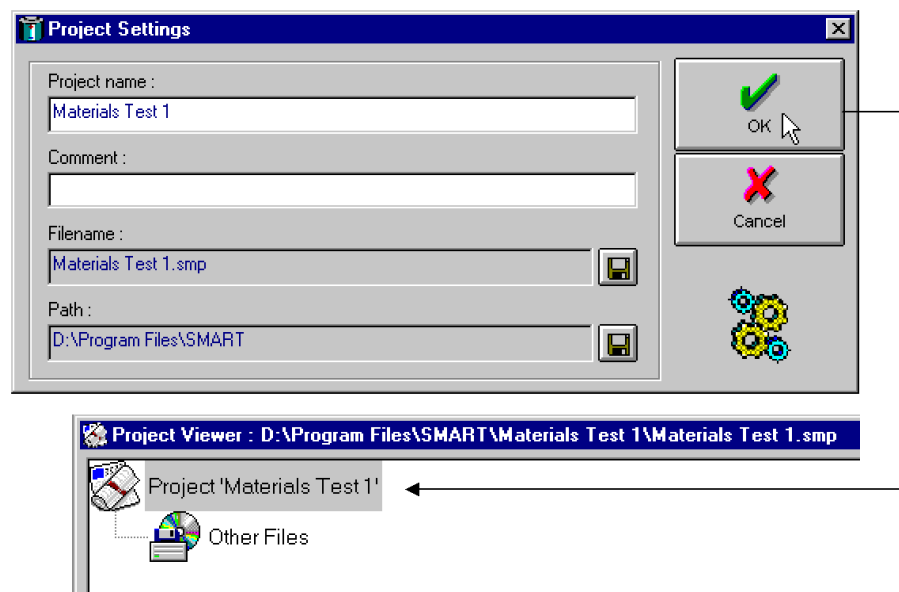
The other two options on the initial ***Welcome to SMaRT*** panel allow you to ***Create a new project*** or ***Open your previous project***. Use these when you are familiar with the operation of SMaRT, to enable you to set up your project directly.

2.1 CREATING YOUR PROJECT

To start creating your first project, click on this button:



This reveals the **Project Settings** window¹. Initially, all you do here is give your project a name and say where you want the project directory saved. Once projects have been created, you will have the option of creating a new project or recalling an existing one. When you click on **OK** to assign your project name, the **Project Viewer** appears:



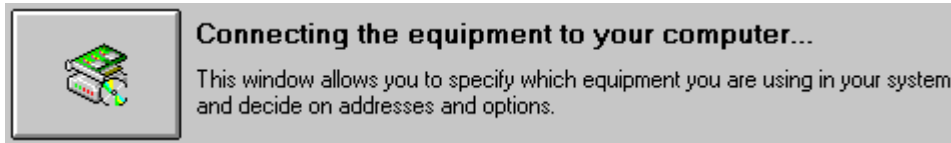
Your project will consist of a group of files related to one or more experiments with a common aim. This will be to do impedance tests on a particular type of material, for instance. In creating a project, you open a **project directory** and assemble in this the files that you wish to use in the project. The header icons that appear in the **Project Viewer**, as shown in the example above, indicate the types of files that you will use. At this time, there are no files in the project directory. Icons representing these files will appear as soon as you create them, or, later, add them from other projects.

Full details of how to create and manage a project appear in Section 4 of the User Guide.

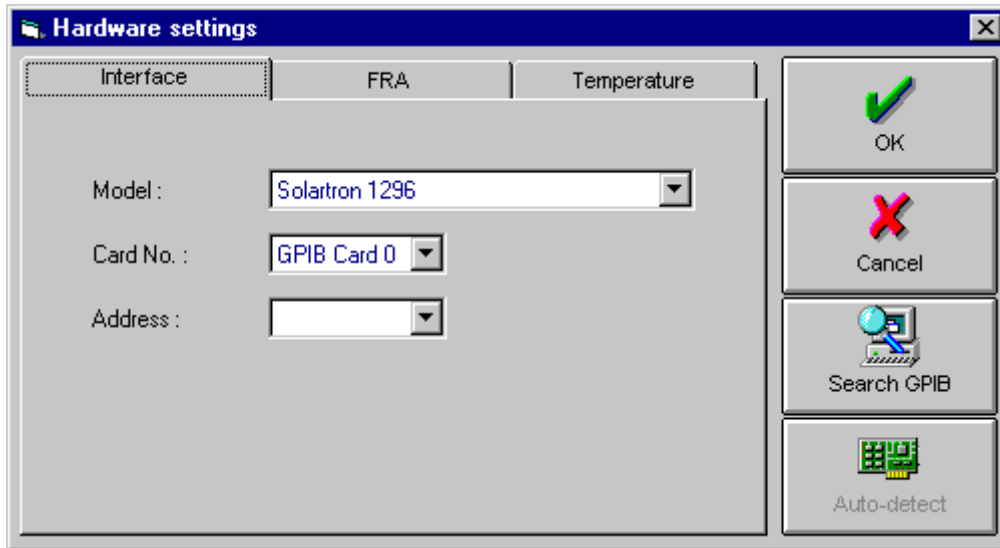
¹ You can access this window directly, by selecting **Create a new project** on the initial **Welcome to SMaRT** panel.

2.2 SPECIFYING THE SYSTEM HARDWARE

To specify the hardware you are going to use for your experiment, click on this button:



This reveals the *Hardware settings* window:

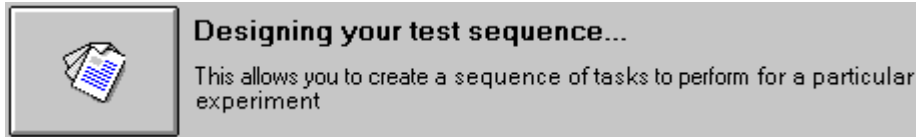


On this, you specify the type of Interface, FRA and Temperature Controller you are going to use and their addresses and options.

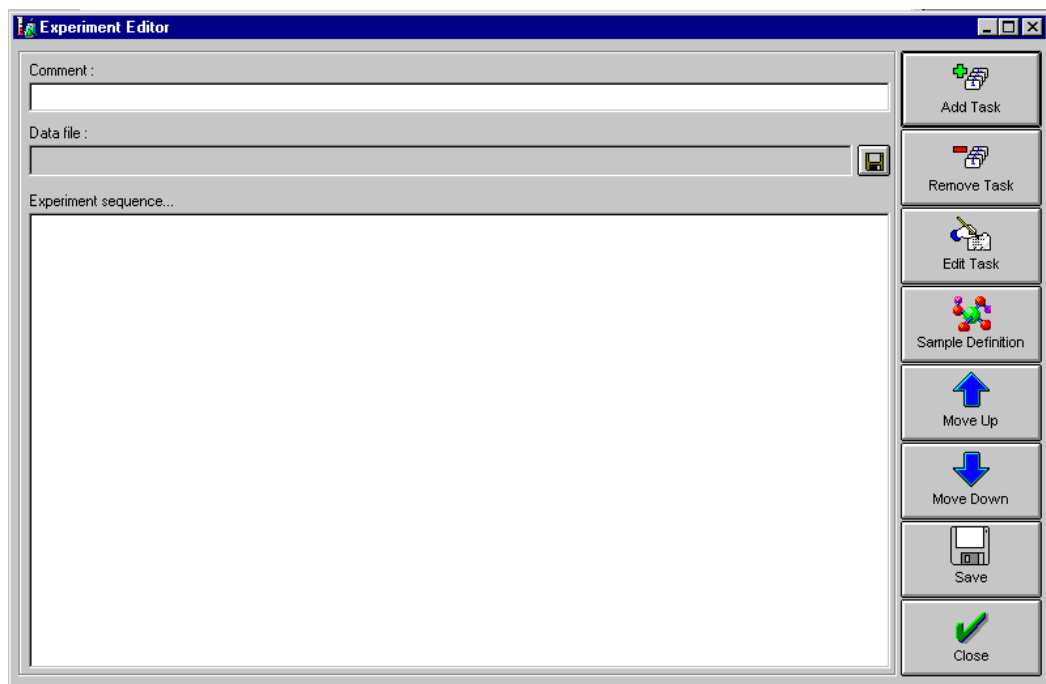
Full details of how to specify the system hardware appear in Section 3 of the User Guide.

2.3 DESIGNING YOUR TEST SEQUENCE

To start designing the test sequence for your experiment, click on this button:



This reveals the *Experiment Editor* window:

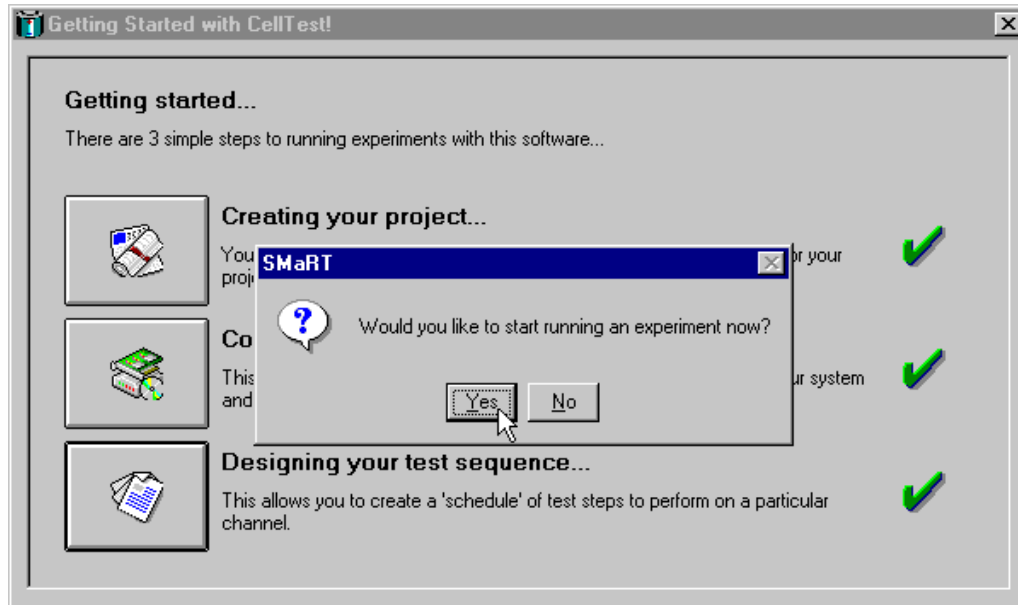


Here you add a series of tasks, which define the experiment parameters. For example, you can enter the frequency and level of the ac source, the dc bias, and the measurement parameters. You can also set up a sweep, of the frequency value for example, or you if you wish can set up several nested sweeps. Full details appear in Section 5 of the User Guide.

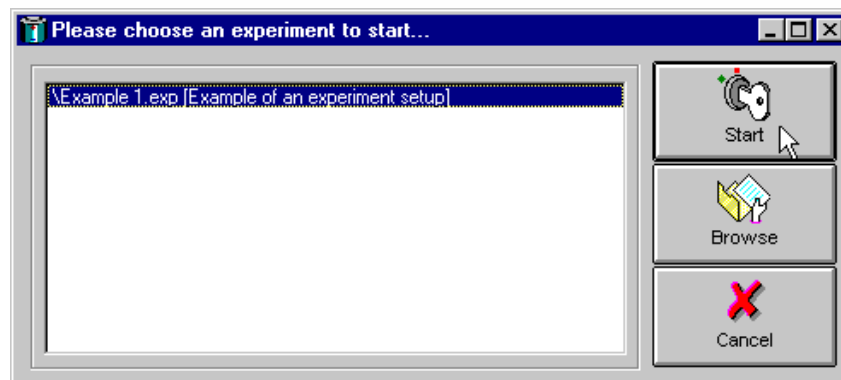
Included in the experiment set-up, is the sample definition. Here you enter the information that enables the 1296 to compute the relative permittivity of the sample to be measured. The information you enter can be either the cell dimensions or the empty cell capacitance.

2.4 STARTING AN EXPERIMENT

As you complete each step of the Getting Started procedure, a green tick appears against it and, on completion of the last step, a message prompts you to start running the experiment:



Selecting Yes reveals the window entitled, “*Please choose an experiment to start...*”



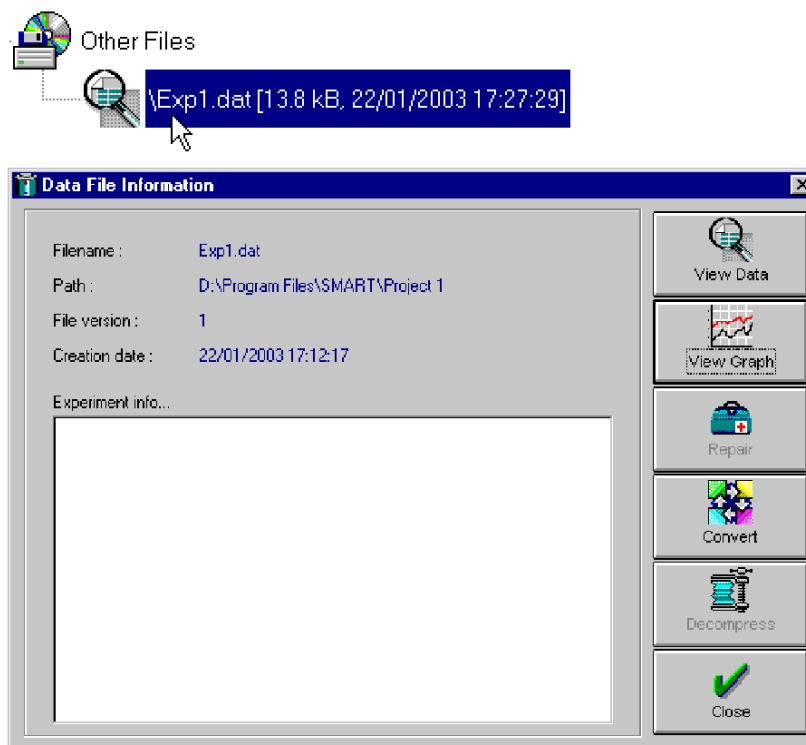
All you do here is select an experiment from the list and click on the **Start** button. The experiment is then executed.

Initially, of course, there will be just the one experiment to choose from, but once several experiments have been created, you can choose any one of them from the list displayed. The browse button enables you to search for and run experiments other than those listed under your current project.

Full details of how to run experiments appear in Section 6 of the User Guide.

2.5 VIEWING THE RESULTS OF YOUR EXPERIMENT

On completion of your experiment, an icon representing the data file appears on the Project Viewer. Double-click on this icon to reveal the Data File Information window:



Here, you can opt to view your data as a graph or in tabular form. You can also repair damaged data files, or convert them to .CSV format for export to other applications such as Excel.

Full details of the ways you can view your data appear in Section 7 of the User Guide

3. Specifying the System Hardware

The system hardware includes the Impedance/Dielectric Interface and any other equipment that may be required, such as option cards, FRAs and temperature controllers.

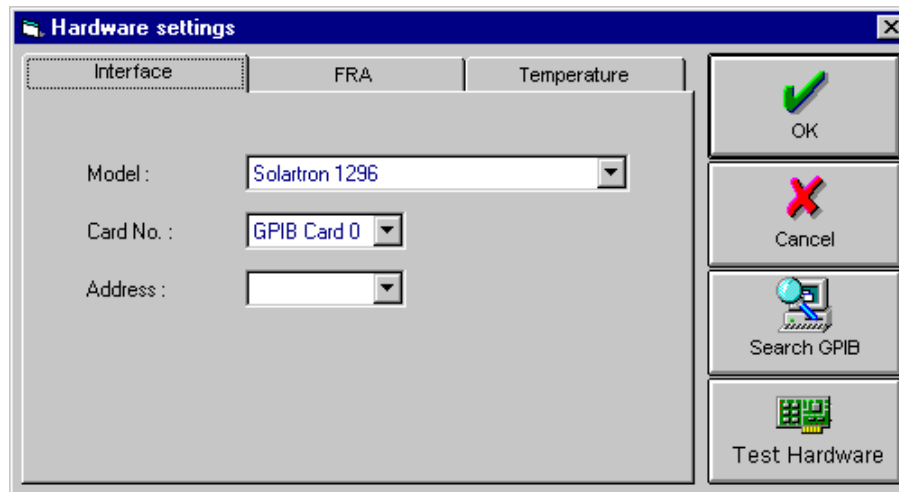
3.1 ENTERING THE HARDWARE SET-UPS

To be able to use an item of hardware you must first assign it to the system software. The procedure is:

1. Click on the **Add/Remove hardware** icon in the toolbar (or, in the **Tools** menu, click on **Hardware settings...**).



This displays the **Hardware Settings** window:

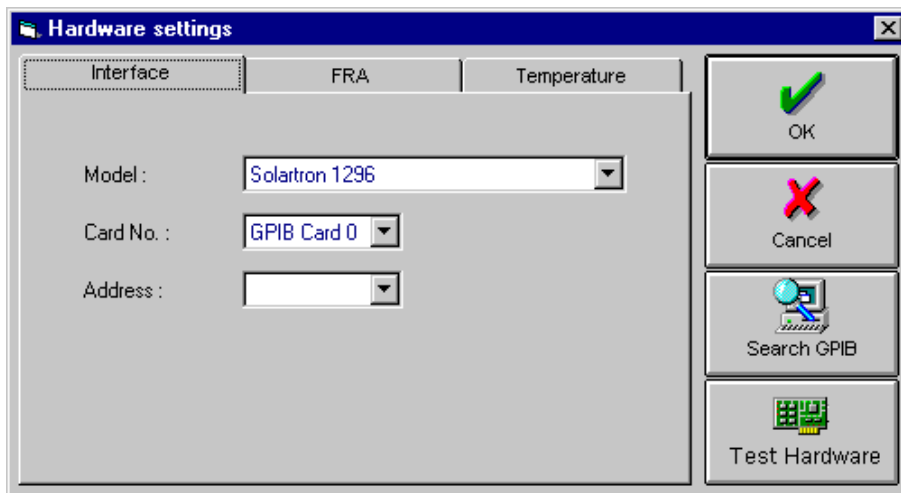


2. On the three tabs of the **Hardware Settings** window, select the Interface, FRA, and Temperature Controller you intend to use in your project. For each of these you will also need to define the port address and type of connection used. See Sections 3.1.1 through 3.1.3 for the details

The GPIB settings apply to the interface or Frequency Response Analyser. If in doubt as to whether or not you have entered the correct GPIB address, click on **Search GPIB**. The message, **Searching for hardware on the GPIB... please wait**, is then displayed, followed by the result of the search. If necessary, correct the GPIB address. The serial port settings apply to Temperature Control Units. The **Test Hardware** facility enables you to check the response of an instrument when addressed.

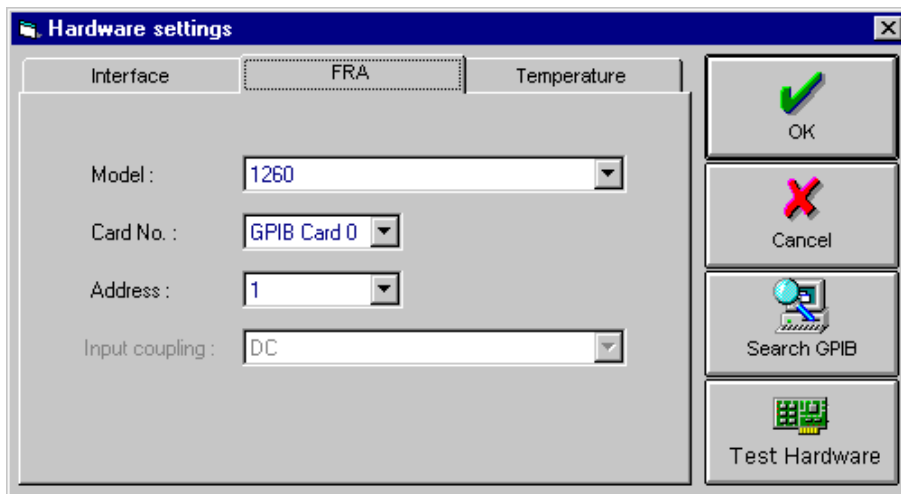
3. Click on **OK** to register the hardware set-up.

3.1.1 SETTING THE DIELECTRIC INTERFACE



- Model: 1296, 1294 or none (1260 stand alone mode)
- Card No. Identifies the system controller GPIB card in the PC
- Address: GPIB address of the Dielectric Interface (1 through 8)

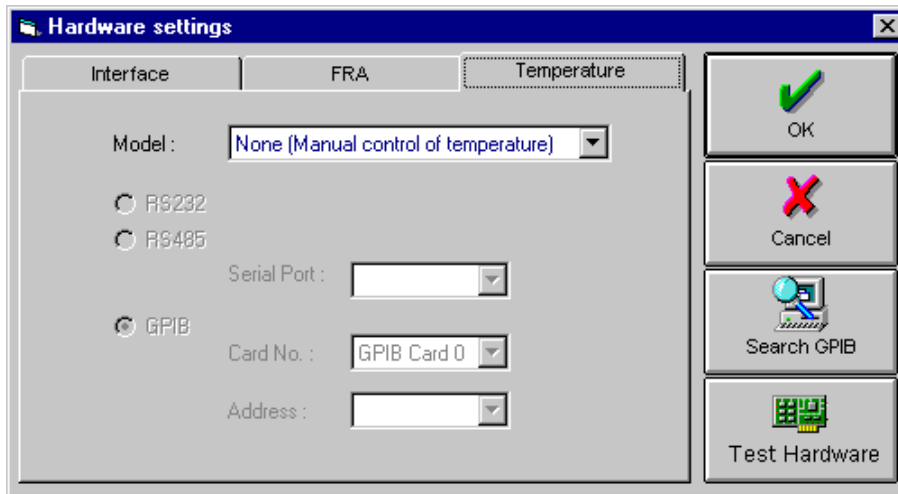
3.1.2 SETTING THE FREQUENCY RESPONSE ANALYSER (FRA)



- Model: Solartron 1260, 1250, 1253, or 1255, or Hewlett Packard HP4192 (See Appendix E for details.)
- Card No. Identifies the system controller GPIB card in the PC
- Address: GPIB address of the FRA (1 through 8)

Note: The FRA uses two addresses the even one shown in the Address box and the address that follows it. A warning is displayed if you attempt to enter the second FRA address for any of the other instruments in your system.

3.1.3 SETTING THE TEMPERATURE CONTROLLER



Model: 'None' –an instrument controlled from outside the system, as selected above– 'Lakeshore 340', 'Oxford Instruments ITC503', or 'Eurotherm 2000 Series'.

If you choose to adjust the temperature manually, SMaRT prompts you to do this whilst the experiment is running. If no temperature control is required then select manual mode –and select fixed, with no prompt in experiment set-up.

Note: The temperature controllers Lakeshore 340 and Oxford Instruments ITC503 are set up, prior to being operated, from their respective set-up files. These files, which are supplied with the impedance measurement software, are Lakeshore.set and oi_setup.set. Guidance on modifying the content of these files for your particular application is given in Appendix C of this manual.

Address: For Lakeshore 340, Oxford Instruments ITC503, or any other temperature control device, you must enter an RS485 address.

Set-up File: For Lakeshore 340 and Oxford Instruments ITC503, you must select a set-up file. For any other type of temperature control device, you must select a set-up file and a DLL file.

4. Creating and Managing a Project

4.1 PROJECT FILES

A project contains the files of a common category of experiments. This allows you to manage the files related to a particular project, which keeps everything nice and tidy. To identify a project you enter a title in the project name box. You can enter a broader description of the project in the “comments” box.

The file types contained in a project are:

- Schedule - holds a test sequence: charge, discharge, etc.
- Experiment file - holds a definition of the hardware to use for an experiment and the name of the data file to use.
- Data file - holds the data generated by an experiment.
- Graph file - holds the settings of a specific graph and reference to the data file holding the data you wish to plot.
- Report file - holds the text that summarises a specific experiment, i.e. general information that you may wish to print and file.
- Other files.

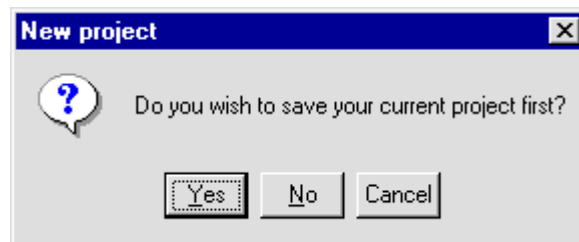
4.2 CREATING A PROJECT

The basic procedure for creating a project is:

1. Click on the **Create a new project** icon in the toolbar (or, in the **File** menu click on **New → Project...**).

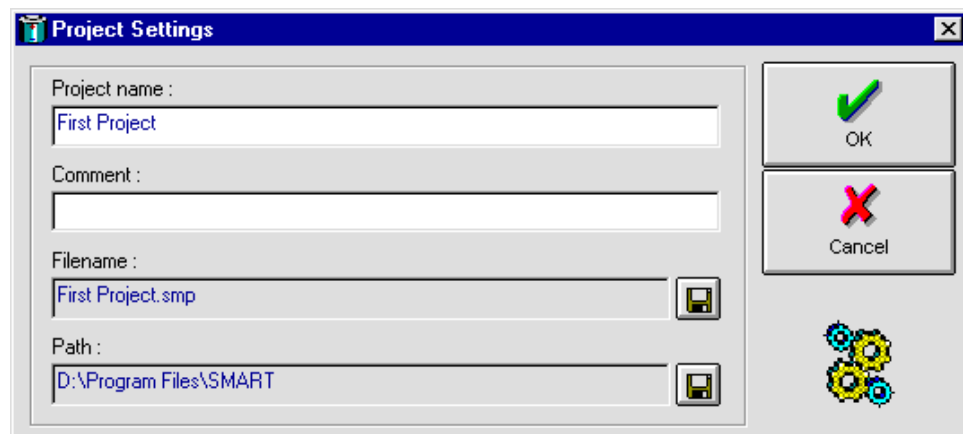




The following message is then displayed:

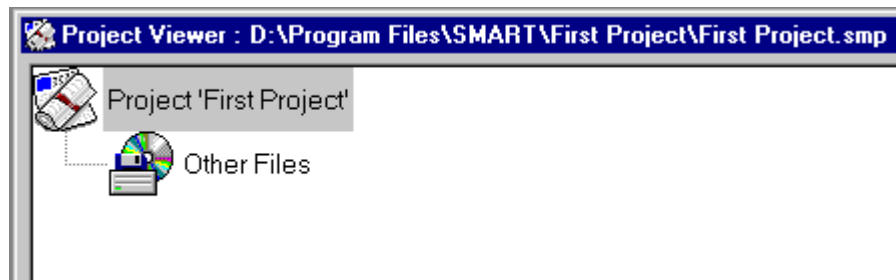


This gives you the option of saving your current project (should you have created one prior to starting another). 'Yes' saves the current project – 'No' discards it. In either case, the **Project Settings** window appears. 'Cancel' aborts the creation of a new project.

The **Project Settings** window appears thus:



2. Enter a name for your project. This name will appear on the project map. If required, enter a brief description of the project under **Comment**.
3. By default, the project name is adopted as the filename, but you can modify this as you think fit. To set a filename, click on the relevant  button. Beware of using illegal characters in the filename. The directory path shows where the directory is to be stored: this too can be modified, by clicking on the other  button.
4. To save the project, click on **OK**. The **Project Viewer** then appears:



Icons representing the various files in the project will appear as these files are saved. **Note that** there are other project 'tree' styles. See Section 8.3.

4.3 MANAGING A PROJECT

The thing to remember when managing a project is that it is simply a group of related files (experiments, schedules, data files, etc).

The four aspects of project management are:

- Loading:** When you load a Project, you are making available the files relating to a cell test application. Only one Project can be loaded at a time.
- Adding files:** If you wish, you can add new files or other components to your current Project. You can also access and use the files of other Projects.
- Backing up:** As a precaution against experiment data being lost, it is possible to back up the Project component files. You can select daily backups, at a selected hour, or periodic backups, at defined periods.
- Saving:** Saving a Project retains the grouping of the experiments, schedules, etc. of your current project, but it does not back up the component files. Omitting to save, or deleting a project, does not lose the component files, only the reference to them.

The way in which you deal with each of these aspects of project management is described in sub-sections 3.3.1 through 3.3.4 below.

4.3.1 LOADING A PROJECT

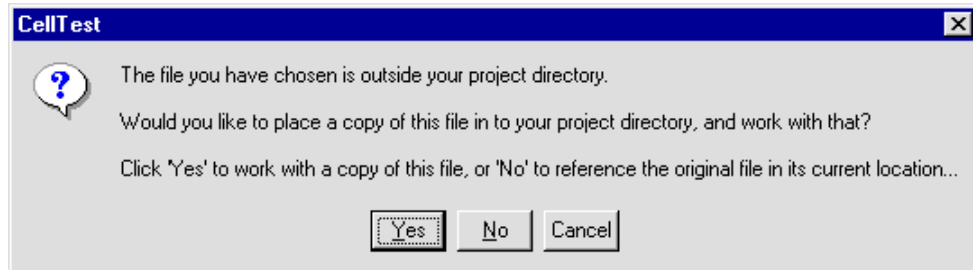
As an option, you can choose to have your previous Project automatically loaded when the application starts up. If you don't select this option then you have the choice either of creating a new project or of opening a previously saved project file. A list of recent projects is accessible from the **File** menu.

Note that only one Project will run at any one time. An attempt to load a project while another is running presents you with the choice of saving or discarding the present Project. Should you discard a project, you do not lose the files it contains –only the project 'shell'. If you wish to delete the files as well, you must do this separately.

4.3.2 ADDING FILES FROM OTHER PROJECTS

To add files from other projects, click on **Add Files to Project...** in the **File** menu. Then **Choose a File to Add** and **Open** it.

Should you have set the message option '**Suggest file copy when adding external files to project?**' (Section 10.4) you will see the following message:



Select **Yes** to copy the file to your current project or **No** to access the original file. The first option is useful for creating one file from another, when you can leave the original file intact. You could perhaps use the second option where commonly used files are stored centrally and where any changes made are with the knowledge and approval of all users.

In addition, you can 'drag and drop' files into your project, using the Windows File Explorer. These files are always referenced in their original location, regardless of the above option.


4.3.3 SAVING A PROJECT

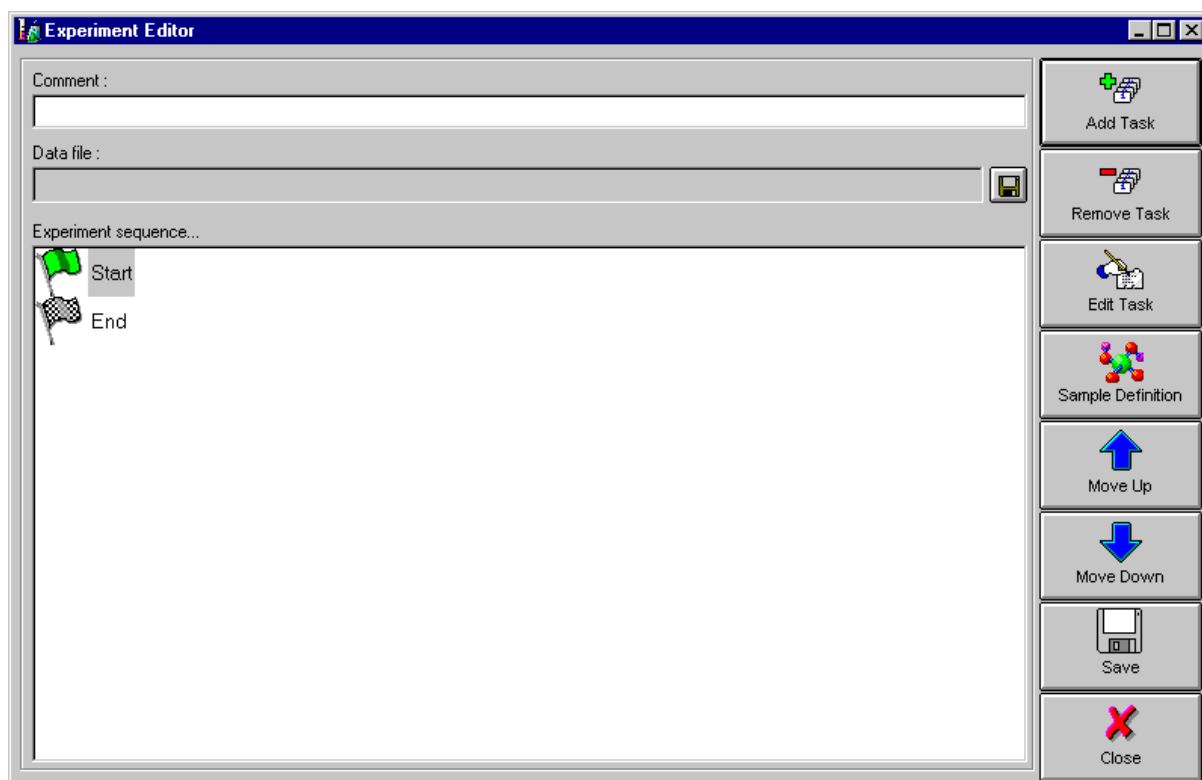
Saving a Project retains the grouping of the experiments, schedules, etc. of your current project. Declining to save a Project deletes the grouping, but the individual components remain available for use in other Projects. Similarly, removing a component from the project map does not delete the component itself, only the reference to it. If you wish to delete the component itself, you must do this separately, by clicking on **Delete**: you must delete a file, for example, from the relevant project directory.

5. Creating an Experiment

5.1 USING THE EXPERIMENT EDITOR

To define an experiment you must specify the way in which the 1296 Dielectric Interface is to drive the sample during the experiment. You must also specify the data file to be used for the experiment results.


To create an experiment you use the *Experiment Editor*. You can access this either by clicking on the  icon in the toolbar or by selecting **New** → **Experiment** in the **File** menu:

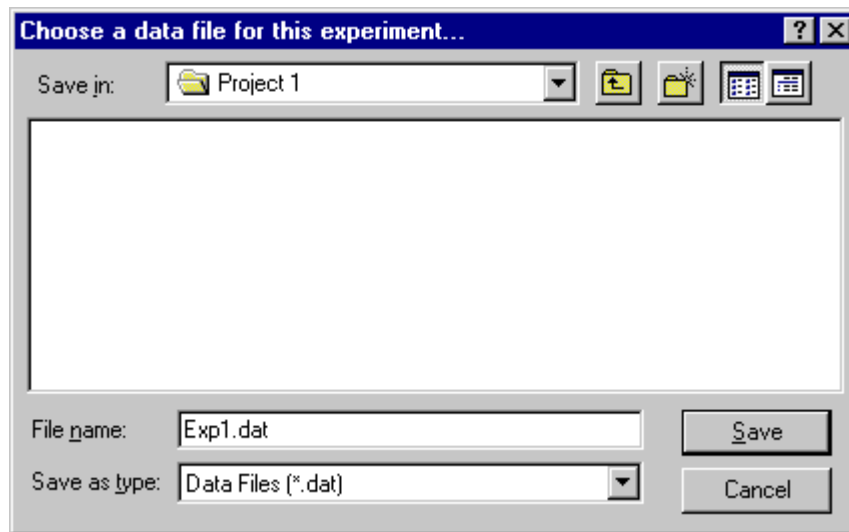


At the top of the Experiment Editor window, you can enter both a comment and the filename for the experiment data. To enter the comment (30 characters maximum) you simply type it into the text box:



When you save the experiment, the comment will appear against the relevant experiment icon on the *Project Viewer*.

To define the filename for the experiment data, click on the  button and use the window entitled **Choose a data file for this experiment...**

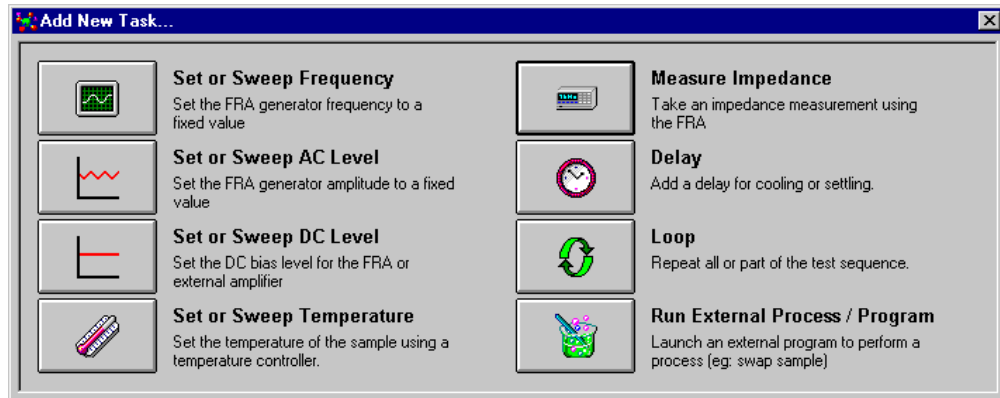


When you save this file name, it appears under **Data file** at the top of the **Experiment Editor** window. When you run the experiment, a data file icon, thus identified, appears under the relevant experiment icon on the **Project Viewer**.

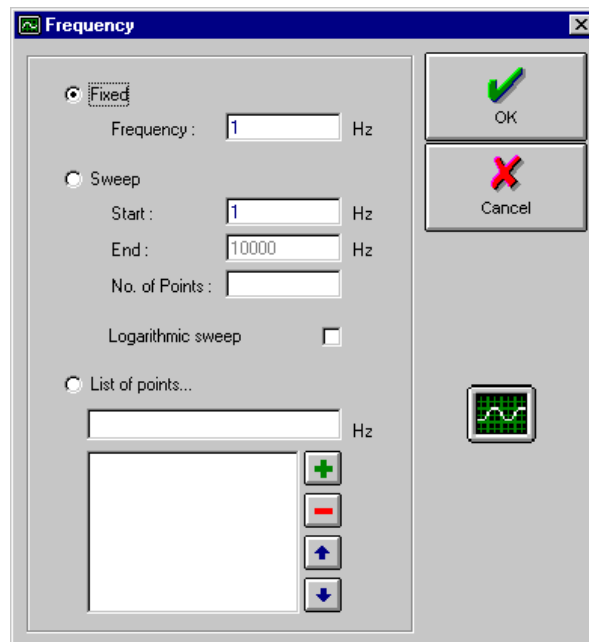
5.2 CREATING AN EXPERIMENT



To create an experiment, you add tasks by using the **Add Task** button. This button opens the **Add New Task** window, which contains a set of buttons that point to all the tasks you will need to perform:



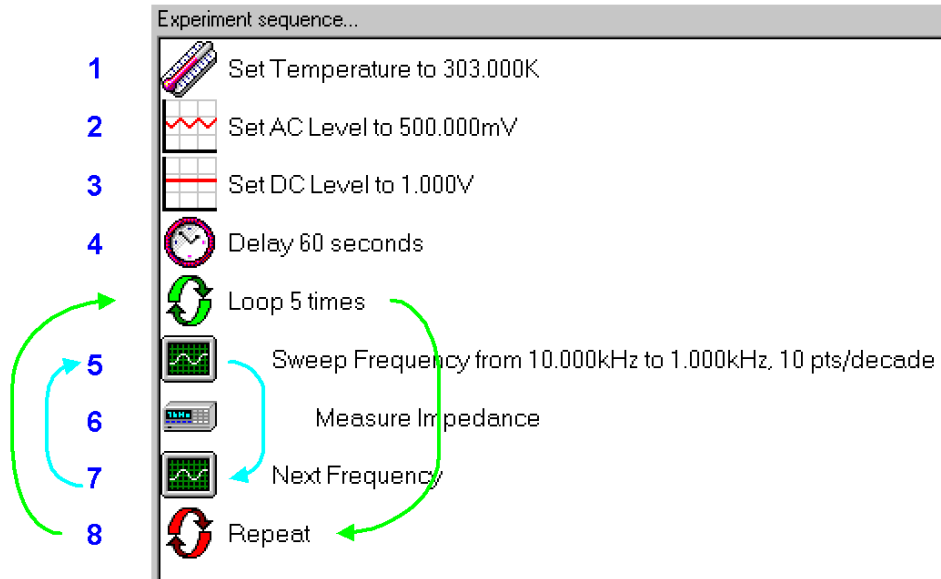
Each button shown here brings up a window for setting a specific experiment task. Defining the measurement frequency, for example:





In this particular window, you can set a fixed frequency for the measurement frequency. Alternatively, you can set the start and end frequencies of a linear or logarithmic frequency sweep. You also have the option of setting a list of randomly defined frequency points.



Similar windows exist for setting the ac and dc levels and temperature. For the 1294, you can also select the **Voltage** or **Current** generator mode. For dc level and temperature, you have the option of prompting for manual control at each point in the sweep.


As you define each task, an icon representing it appears in the **Experiment Sequence** diagram. For example:




The order in which tasks are shown represents their order of execution. In the sequence above, task 1 sets the sample temperature to 303°K (30°C), and tasks 2 and 3 set the ac and dc levels of the FRA generator output to 500mV and 1V respectively. Task 4 then imposes a settling delay of 60 seconds. Following this, tasks 5 and 7 sweep the frequency of the FRA generator output from 10kHz through to 1kHz and with Task 6 making a measurement at each frequency point. Task 8 then repeats the swept frequency measurements five more times.



Use  or  to change the order as required.

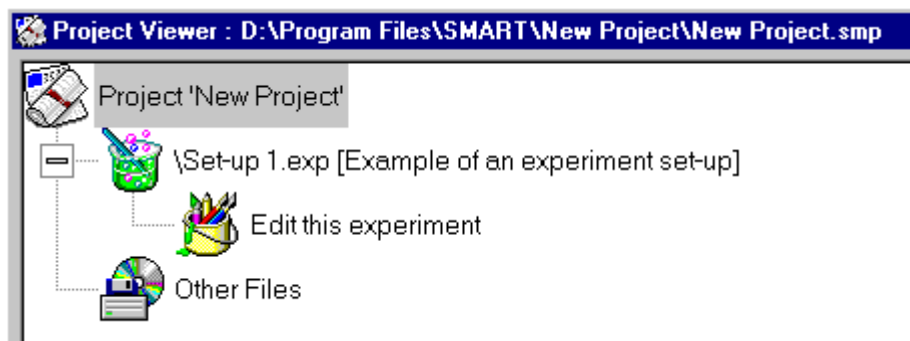
Note the two loops in the above sequence. The outer (green) loop is implemented by inserting a loop task. You can set this to repeat all or part of the experiment sequence a defined number of times. You select the tasks to be repeated by moving the loop start icon  to precede the first task in the repeat sequence and the loop end icon  to follow the last.

To edit an existing experiment task, select  to reveal the task set-up window. Reassign the task parameters and **OK** them, same as during the initial setting up.


To delete an unwanted task from the experiment, select the task icon and  then

Having set up all the experiment tasks required, and defined the sample,

select  to save the experiment. The experiment icon  then appears on the **Project Viewer**.




Note the comment: [Example of an experiment set-up]. This is the one entered at the top of the Experiment Editor. You specify the experiment filename, in this example "Set-up 1.exp", when you save the experiment.

To edit an existing experiment, double-click on  to reopen the Experiment Editor. You can then edit, add and/or delete tasks, before saving the experiment, either as an updated version or as a completely new one.

5.3 DEFINING THE SAMPLE

The Sample Definition window enables you to enter the empty cell capacitance of the sample, or the dimensions from which this may be derived. The empty cell capacitance enables the 1296 to compute the relative permittivity of the sample.



To enter the sample definition, click on the  button. This reveals the **Sample Definition** window:

 A screenshot of the "Sample Definition" dialog box. The window has a blue title bar with the text "Sample Definition" and a close button (X). The main area contains three radio button options:

- Circular Cross Section: This option is selected. It includes three input fields: "Sample Diameter : 10 mm", "Sample Area : 78.54 sq mm", and "Sample Thickness : 0.5 mm".
- Non-Circular Cross Section: This option is unselected. It includes one input field: "Sample Area : 78.54 sq mm".
- Empty Cell Capacitance: This option is unselected. It includes one input field: "Empty Cell Capacitance : 1.39 pF".

 On the right side of the dialog, there are two buttons: "OK" (with a green checkmark icon) and "Cancel" (with a red X icon). Below these buttons is a colorful molecular model icon.

You can define a sample in either of two ways: enter the sample dimensions and let the 1296 compute the empty cell capacitance; or enter the capacitance value direct. The first method is suitable for a sample for which you know the dimensions, but cannot remove the dielectric. The second method is suitable for samples of an irregular shape: if you do not know the empty cell capacitance, but you can remove the dielectric, the 1296 can measure the capacitance for you.

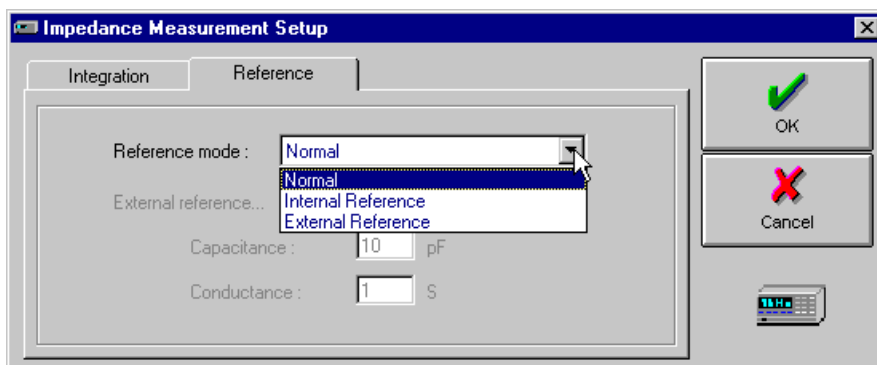
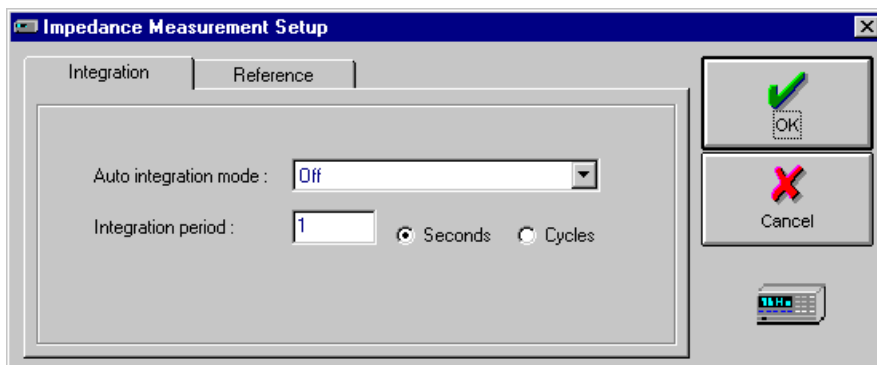
5.4 SETTING UP A MEASUREMENT

The 1296 unit offers two methods of making an impedance measurement - *normal* or *reference*. With the *normal* method, a single direct measurement is made. The results that this gives are accurate at low frequencies, but errors due to stray impedances can arise at high frequencies. To avoid this problem, you can use the *reference* method. With this, two measurements are made: one of the sample and one of a reference capacitor of comparable value. From the ratio of the two results and the reference capacitance value the 1296 computes the true sample value.

To reject any noise that may be present in the sample signal the 1296 allows you to select integration of the Analyser input.

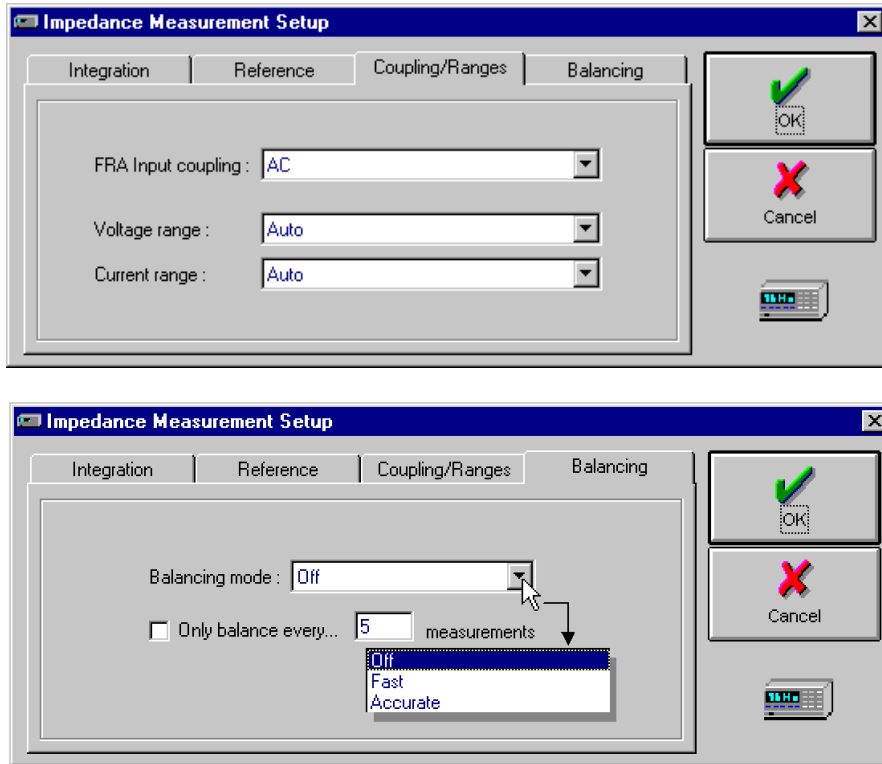
The measurement method can be set at various points in the experiment sequence, to suit the parameters of the sample stimuli –which are the parameters set for the generator output. For example, you can use the reference method for high frequency measurements and the normal method for low frequency measurements. These measurement methods are discussed in more detail below.

To set the measurement method, simply add an **Impedance Measurement Set-up** task to the experiment sequence. With the 1296 selected as the hardware interface, the window for this has two tabs, one for selecting the analyser integration and the other for selecting the measurement method:



The use of these settings is described in Sections 5.4.1 and 5.4.2.

Alternatively, with the 1294 selected as the hardware interface, two further tabs appear:

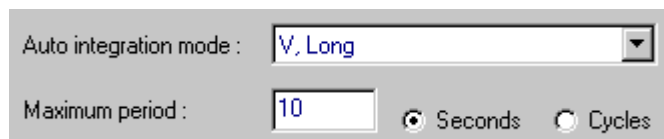


One of these is for coupling and ranging of the FRA input and the other is for generator balancing. The use of these settings is described in Sections 5.4.3 and 5.4.4.

5.4.1 SELECTING THE ANALYSER INTEGRATION (1294/1296)

Noise that is present in the signal applied to the Analyser can be rejected by a process known as signal integration. This process averages the signal over a whole number of cycles, which narrows the measurement bandwidth and thus increases the signal to noise ratio. The more a signal is averaged the longer it takes to get a result, so there is a trade-off between the accuracy you require and the measurement speed.

The measurement set-up defaults to a fixed Integration Period, with auto integration off. If you wish to use a fixed integration period, enter the period you require, in seconds or cycles, in the Integration Period box. Select cycles or seconds by clicking on the relevant option. If you know what accuracy you want in your results, but you don't know how much integration is required to obtain this then a possible solution is to use Auto Integration:



Auto Integration averages the signal until the standard deviation reaches a target value. Details of the standard deviations obtained are given in the relevant FRA manual. To avoid excessive measurement times, you assign a Maximum Period, in seconds or cycles.

When you enter a value in seconds the 1296 or 1294 always selects the most appropriate number of whole cycles. You must specify at least one cycle for the Analyser to operate.

5.4.2 SELECTING THE MEASUREMENT METHOD (1296)

The measurement set-up defaults to normal measurement. To select the reference method, click on internal or external reference in the reference mode menu.

By adding an Impedance Measurement Set-up task at two different points in the measurement sequence, you have the option of using the *normal* method for measurements below a specified frequency and the *reference* method for measurements at frequencies above. This is worth doing when you can obtain the same accuracy with either method over the lower part of the frequency range. The normal method makes only one measurement per result and this saves time.

The frequency below which you use the normal measurement mode depends on the sample and therefore you must find this frequency by experiment. The easiest way to do this is to make two frequency sweeps, one using the normal method and the other using the reference method. Compare the results on a graph and note the frequency at which the traces diverge. Above this frequency, you should use the reference method. Below it, you can use the normal method. You should then obtain a consistent accuracy throughout the sweep, for this particular sample.

5.4.2.1 Selecting the Reference Capacitor

For the internal reference, the 1296 unit automatically selects a capacitance that is comparable to that of the sample. If you wish to use an external reference, however, you must connect a suitable capacitor to the 1296 unit yourself. You must also enter the capacitance and conductance values of this capacitor, on the **Reference** tab of the **Impedance Measurement Set-up** window.

If you use a wide range of frequencies, you may need to use several different values for the external reference capacitor to maintain measurement accuracy. To do this you will need to conduct a series of experiments, for each experiment using a suitable reference capacitor.

5.4.3 ANALYSER INPUT RANGING AND COUPLING (1294)

The voltage sense range of the measurement inputs is used to measure the differential voltage across the sample. The current sense range is used to measure the current through the sample.

The sample current is converted to a voltage in the 1294. This voltage is then measured by the FRA. (Channel 2 of the FRA, which is used for this purpose, is always in auto-range.)

It is best to leave the **Voltage range** and **Current range** settings on 'Auto'. This ensures that an optimum range is selected automatically, in keeping with the signal amplitude. Automatic range selection aims at selecting the least sensitive 1294 range and the most sensitive FRA range possible, thus ensuring the widest possible measurement bandwidth and highest accuracy.

Should you wish to specify the current sense range yourself then you should note the following:

- The current sense ranges are listed as conversion factors: 1V/mA, 1V/10mA, 1V/100mA, and so on. This means that 1V represents 1mA, or 10mA, or 100mA, and so on, depending on the conversion factor that you select.
- In keeping with the conversion factor selected and the current range required for the experiment, you must ensure that the FRA can automatically select a suitable range for the voltage representing the sample current. For example, if the maximum current expected in the experiment is 30mA then you could select a current sense range setting 1V/1mA. The maximum voltage fed to the FRA would then be 30mV, so the FRA should auto-range to the 30mV range.
- Should you wish to obtain a higher voltage to current ratio, then you could, in the above example, select a current sense range setting of 1V/100mA. This would give 300mV as the maximum FRA input voltage and would require the FRA to auto-range to the 300mV range. The more sensitive 1V/100mA range has, however, a lower bandwidth than the 1V/mA range.

Provision is also made for the selection of ac or dc coupling at the FRA input.

5.4.4 GENERATOR BALANCING (1294)

The generator balance facility enables the generator, in voltage drive mode, to compensate for external impedances, such as those contributed by the sample electrodes and test leads. These impedances (represented by Z_1 and Z_2 in Figure 5.1) act with Z_{sample} as a potential divider and cause a common mode signal to appear at the V INPUT LO terminal. A leakage current can thus be made to flow from the V INPUT LO terminal to ground. This current flows through the sample, but, since it does not flow into the GEN LO terminal, it is not measured. This leads to errors in current measurement.

The aim of generator balancing is to avoid a common mode signal at the V INPUT LO terminal. For this purpose, anti-phase versions of the drive voltage signal are applied at opposite ends of the measurement circuit. The relative amplitudes of the opposing signals are adjusted automatically, such that the voltage at the V INPUT LO terminal is minimised.

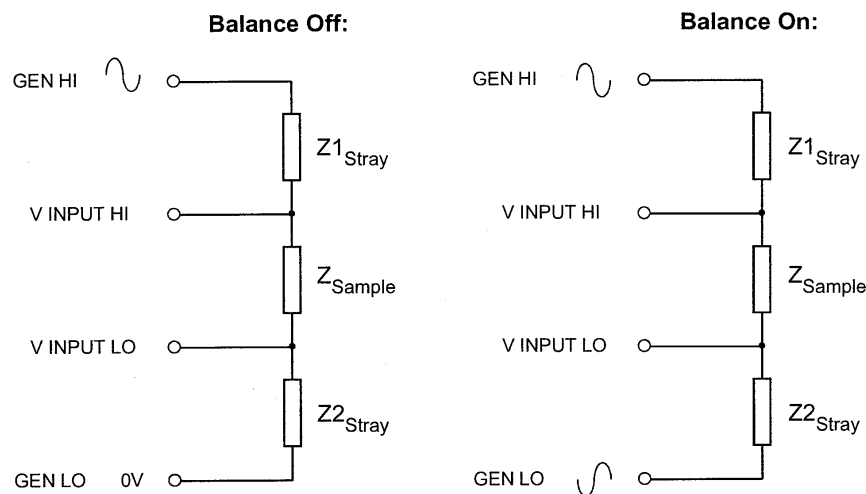


Figure 5-1 Generator balance for four-terminal measurement.

Three balance settings provide for various experiment conditions:

Off, where balance is not required,

Fast where balanced measurements are required, but speed has priority,

Accurate where balanced measurements are required, but precision has priority.

Depending on the rate at which the external impedances are likely to change, you can also arrange for balancing to occur at every n^{th} measurement. This can be used to advantage where measurement precision and speed are both important.

6. Running an Experiment

6.1 STARTING AN EXPERIMENT

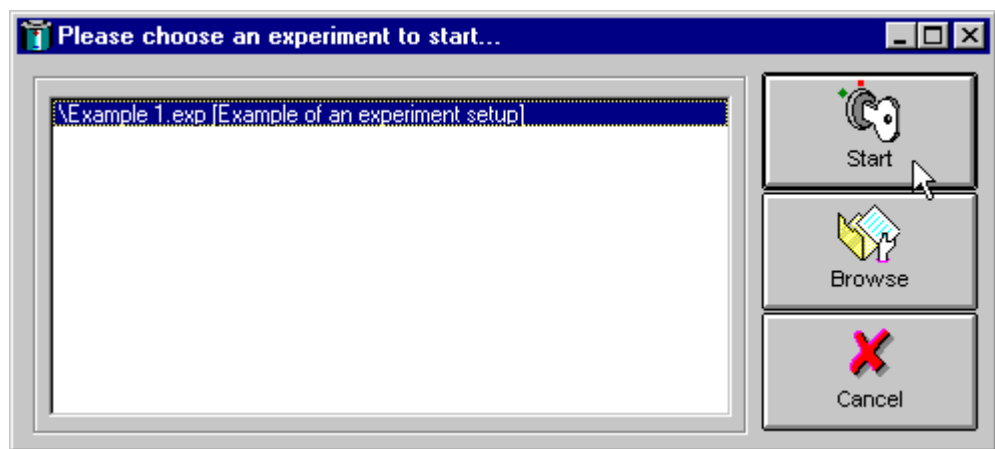
The procedure for starting an experiment is:

1. Click on the traffic light icon...

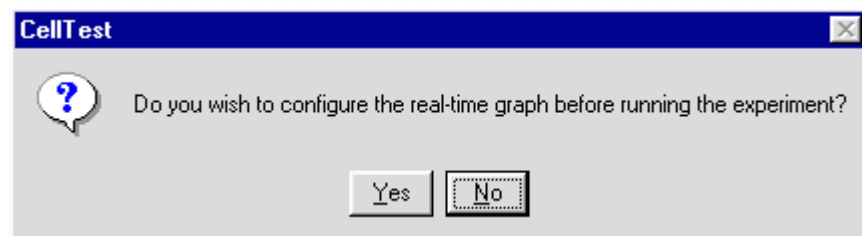


...or on **Start Experiment...** in the **Control** menu.

A window entitled **Please choose an experiment to start...** appears:



2. On the above window, select an experiment then click on the **Start** button:
3. Under **Tools, Options, Graphics** you have the option **Show a new graph when the experiments starts**. (See Section 10.2, 'Selecting the General Options'.) If this option is selected, a message appears, before the experiment starts, giving you the option of configuring the real-time graph of the experiment results:



If you select **yes**, the graph configuration window appears (see Section 9.1). If you select **no**, the experiment results are plotted on a default graph of voltage against time.

Following step 3 (or step 2, if the graph option mentioned in step 3 is not selected) the experiment starts and its progress is shown on the **System Status Grid**.

The grid shows, for each channel in the experiment, the quantities measured and the estimated measurement time left. Stop, run and skip buttons are also included, for each channel. This is demonstrated in the following example.

6.2 STOPPING AN EXPERIMENT

To stop all experiments, click on the  button in the CellTest tool bar or on **Stop** in the **Control** menu. The stop button is intended for emergency stops.

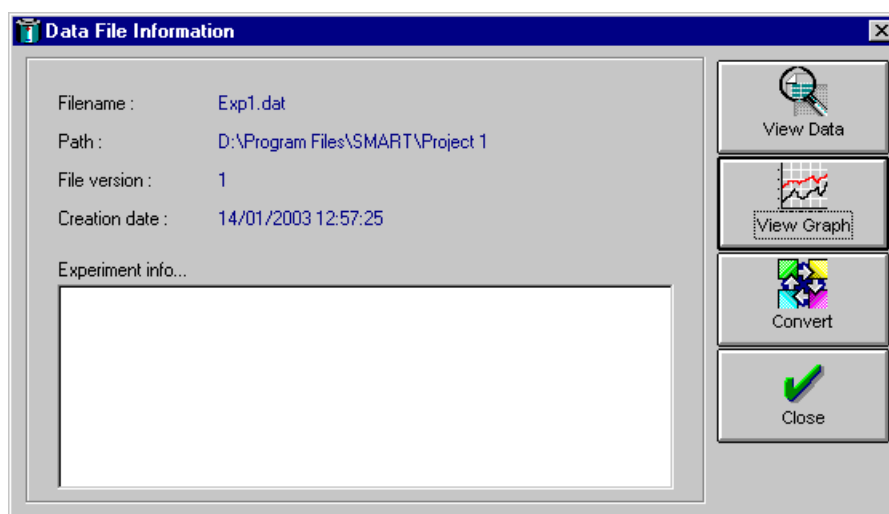
However, as with all computer-controlled systems, you should not rely on this alone. This is because a communications failure may prevent the stop command being actioned. This applies particularly where injury or damage could result from a critical experiment. In such a case, it may necessary to disconnect the ac supply to the unit that is driving the critical experiment.

Sensible precautions should be taken against the possible hazards of a critical experiment –explosion, fire, and corrosive liquids, for example– which could occur before the experiment is brought under control. This could take the form of a complete enclosure or a screen, depending on the degree of danger. Fire prevention should also be considered. All precautions should be taken in accordance with local safety regulations regarding hazards of this nature.

7. Viewing Your Results

7.1 PREPARING TO VIEW

Once you have run an experiment, data is available in the designated data file(s) and a representative data file icon appears under the **Other Files** icon on the **Project Viewer**. Double clicking on the data file icon opens the **Data File Information** window, which contains a summary of the filed data. Here you can select the data you wish to study and how you wish to view it: as **data**, in tabular form, or as a **graph**. You can also **convert** your data before viewing it.



The procedure for viewing a graphs and tables is described in Sections 7.2 and 7.3, whilst file conversion is described in Section 7.4.

7.2 VIEWING YOUR RESULTS ON A GRAPH

For real-time or historic graphs, you configure the presentation in the same way. The only difference between the two is the way in which the data appears. Real-time graphs build up as the experiment proceeds, whereas historic graphs represent filed data already acquired and appear as a complete plot.

To access the graph configuration windows, select **View Graph** on the **Data File Information** window or click on the **Show a graph** icon in the toolbar. This reveals the six graph configuration windows. Sections 9.1.1 through 9.1.6 below explain the use of these. When graph configuration is complete, click on **OK** for plotting to begin. Note that you can **Save** a graph configuration and **Load** a previously saved one.



7.2.1 SELECTING THE PLOT TYPE

On the **Plot Type** window (Figure 7.1), select the type of graph you want. This can be for **Bode** or **Complex plane**. From here, you proceed from left to right, setting up the trace(s), x-axis, y-axis, etc, as explained in the following sections. When you finish configuring your graph, click on **OK** to display it.

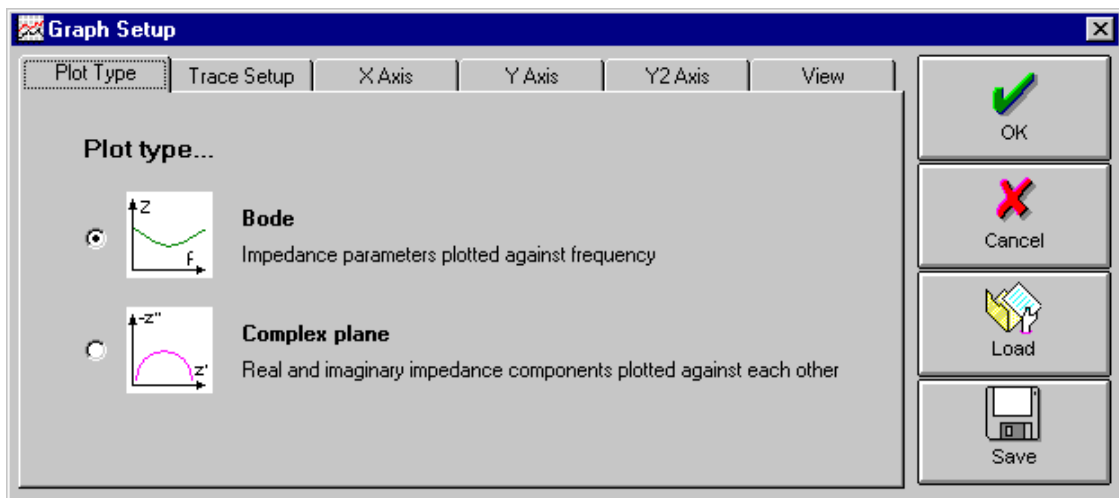


Figure 7-1 Graph configuration: plot type.

7.2.2 SETTING UP THE TRACES

On the Trace Set-up window (Figure 7.2), you assign the data you wish to plot. Up to eight traces, selected from various data sources, can appear on the same graph.

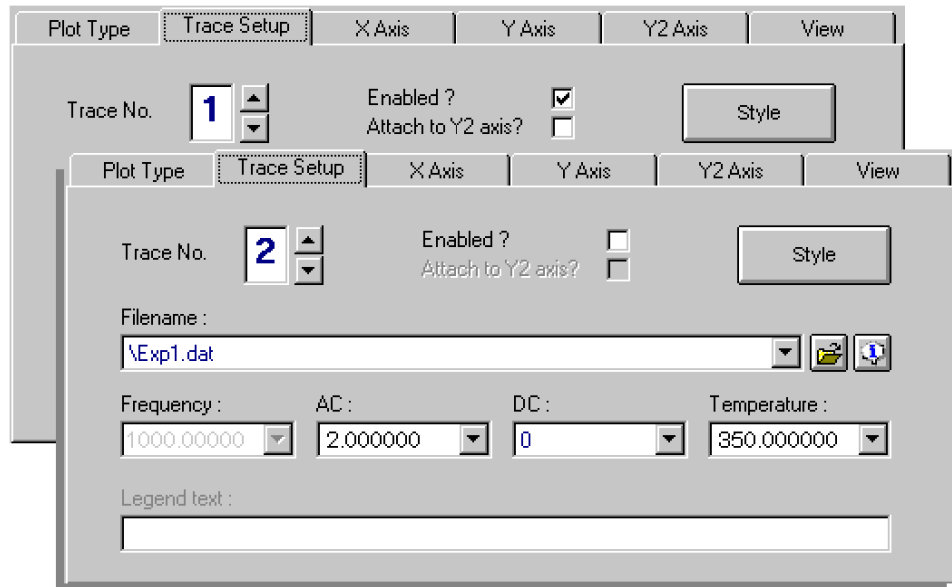


Figure 7-2 Graph configuration: trace set-up.

The procedure for assigning data to a trace is:

1. Select a trace on which to plot the data. Under **Trace set-up**, select a **Trace No.** (e.g. '1') and click to tick **Enabled**. To make it easier to differentiate between several traces on a graph, each trace has a different style, e.g. colour, thickness, etc. If you wish to define your own styles, you can do this in the **Style** window. To plot a trace against the **Y2 axis**, click to tick **Attach to Y2 axis**.
2. Select the **Filename** of the file containing the data you wish to plot.
3. Select the required sweep. To accommodate nested sweeps, the currently selected parameters are presented in four boxes entitled **Frequency**, **AC**, **DC** and **Temperature**. Simply select the values defining the trace you require. A typical example of nested sweeps could be: at each of three temperatures 300°K, 350°K and 400°K, and at each of three ac levels 1Vrms, 2Vrms and 3Vrms, run the frequency sweep 1kHz through 10kHz. Here, you would have 1, 2, and 3 listed under **AC** and 300, 350 and 400 listed under **Temperature**. Selecting '2' and '350' would select the trace for a temperature of 350°K and ac level of 2Vrms.
4. Repeat steps 1 through 3 for each trace you wish to plot.

7.2.3 SETTING UP THE X AXIS

The type of units you can plot on the x-axis depends on the selection you make under **Plot Type** (see Section 7.1.1). By default, the x-axis is in units of **Frequency** for **Bode** plots. You can select alternative units to suit your needs, under **Type**. (See Figure 7.3.) Full lists of the units available appear in Section 1.

To plot the full duration of the data that is available from your selected data source, click to tick **Auto scale**. Alternatively, to plot part of the data, you can define the **Min** and **Max** limits of a fixed scale. Assume, for example, that the full frequency range of your data is 0Hz through 1MHz and you wish to plot the frequency band 100Hz through 10kHz. First, deselect **Auto scale**, by clicking with the left mouse button. Then set **Min** to '100' and **Max** to '10000'. Note that you can also enter the fixed scale values in exponential form, which in the present example would be **Min** = 1e2 and **Max** = 1e4.

To plot your results in the most meaningful way, you can select **Linear** or **Logarithmic** scaling.

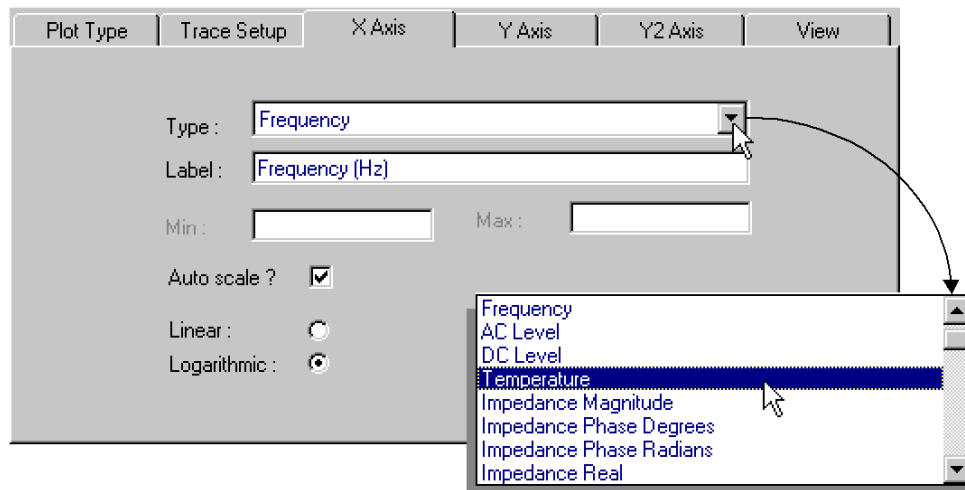


Figure 7-3 Graph configuration: x-axis.

7.2.4 SETTING UP THE Y AXES

The data type you can plot on the y-axis depends on the selection you make under **Plot Type** (see Section 7.1.1). By default, the data types selected for **Bode plots** are Y =magnitude and Y2 =phase. You can select alternative data types to suit your needs, under **Type**. These include impedance real and imaginary, or admittance magnitude and phase for Bode plots. Full lists of the units available appear in Section 1.

Beside each Y-axis, you will see the data type label. By default, the labels are the data type and units. You can enter your own label for each trace under **Label**. This then replaces the default label.

To cover the full range of data available on all selected traces, click to tick **Auto scale**. Alternatively, to plot the amplitudes within a restricted range, you can define the **Min** and **Max** limits of a fixed scale. Assume, for example, that the amplitude of your data ranges between 1kΩ and 100MΩ, and you are interested in amplitudes between 10kΩ and 1MΩ. First, deselect **Auto scale**, by clicking with the left mouse button. Then set **Min** and **Max** accordingly, e.g. '10000' and '1000000'. The precision of your entries should match that of the cursor readout, i.e. to three, six, or nine decimal places (see Section 8.5). Note that you can also enter the fixed scale values in exponential form, which in the present example would be **Min** =1e4 and **Max** =1e6.

To plot your results in the most meaningful way, you can select **Linear** or **Logarithmic** scaling.

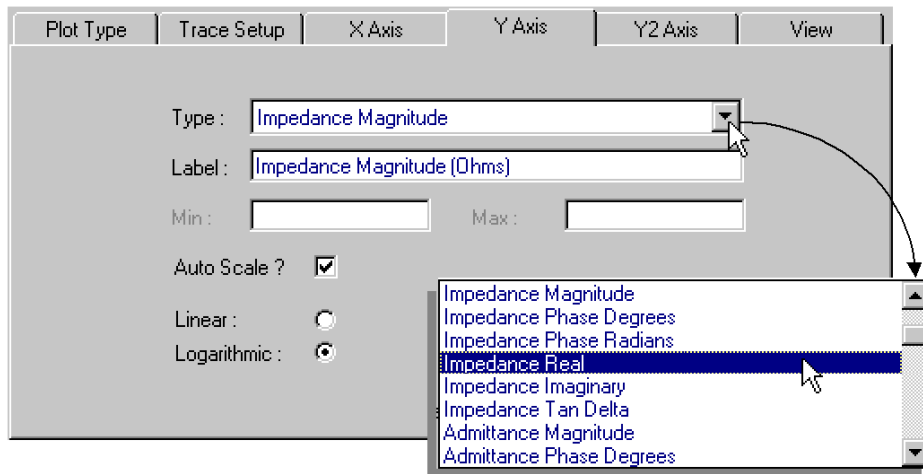


Figure 7-4 Graph configuration: y-axes.

7.2.5 SETTING UP THE VIEW

On the **View** window, you set up the general appearance of your graph.

The **Display** options apply to all graphs. With these, you can:

- Display the grid, or not,
- With the grid displayed, display the minor divisions, or not,
- Display the cross-wire cursors, or not,
- Display the legends (trace labels), or not.

The **Complex plane options** are available only with **Complex plane** selected as the **Plot Type**. These options allow you to:

- Use isotropic scaling (equal scaling on x- and y-axes), or not,
- Invert the y-axis, or not.

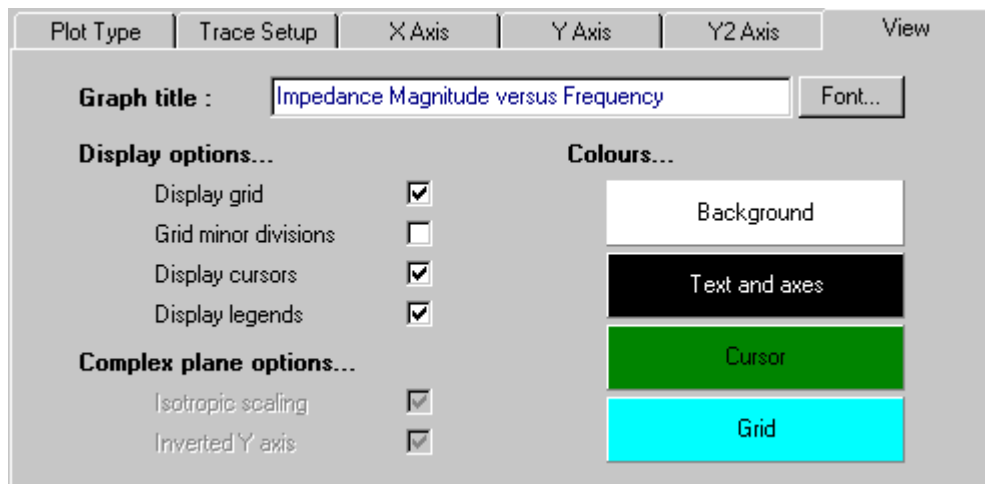


Figure 7-5 Graph configuration: view.

An example of an historic graph appears in Figure 7.6. Note the precision of the cursor reading, which appears at the bottom of the graph frame, in the centre. On the **Options** window for **Graphics**, (see Section 8.5) you can select a precision of three, six, or nine (default setting) decimal places.

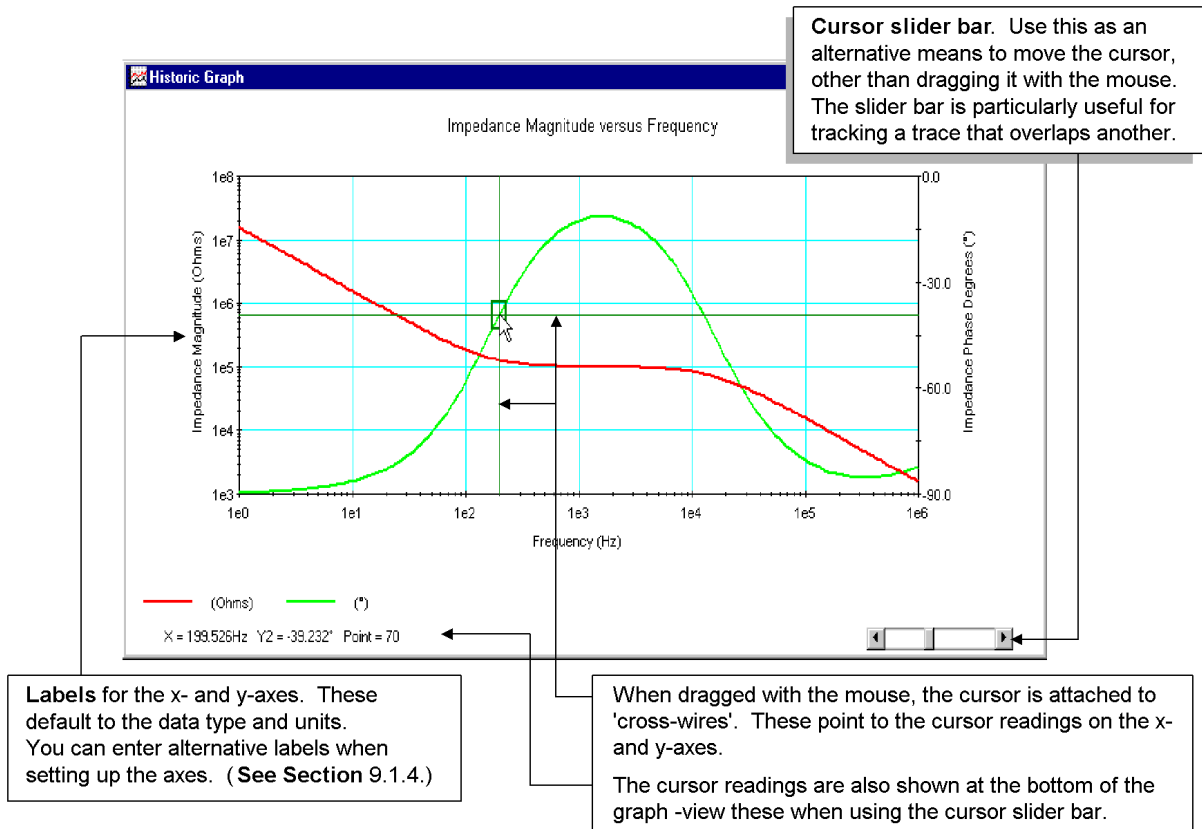


Figure 7-6 Example of a Bode plot.

Before attempting to use the cursor slider bar, (see Figure 7.6) locate the graph cursor by clicking onto the appropriate trace, then hold down the keyboard **Ctrl** key whilst moving the mouse pointer to the slider bar.

7.2.6 ZOOMING IN ON THE GRAPH DETAIL

The cross-wire cursor allows you to zoom in on any part of a plot, on both axes. Simply click and stretch a box over the required area. When you release the left mouse button, the selected area appears as a full graph. See Figure 7.7. Click with the right mouse button to reveal a pop-up menu. This enables you to set up the graph configuration, zoom level and scroll time; to save the graph configuration or load a previously saved configuration; and to export, print or close the graph.

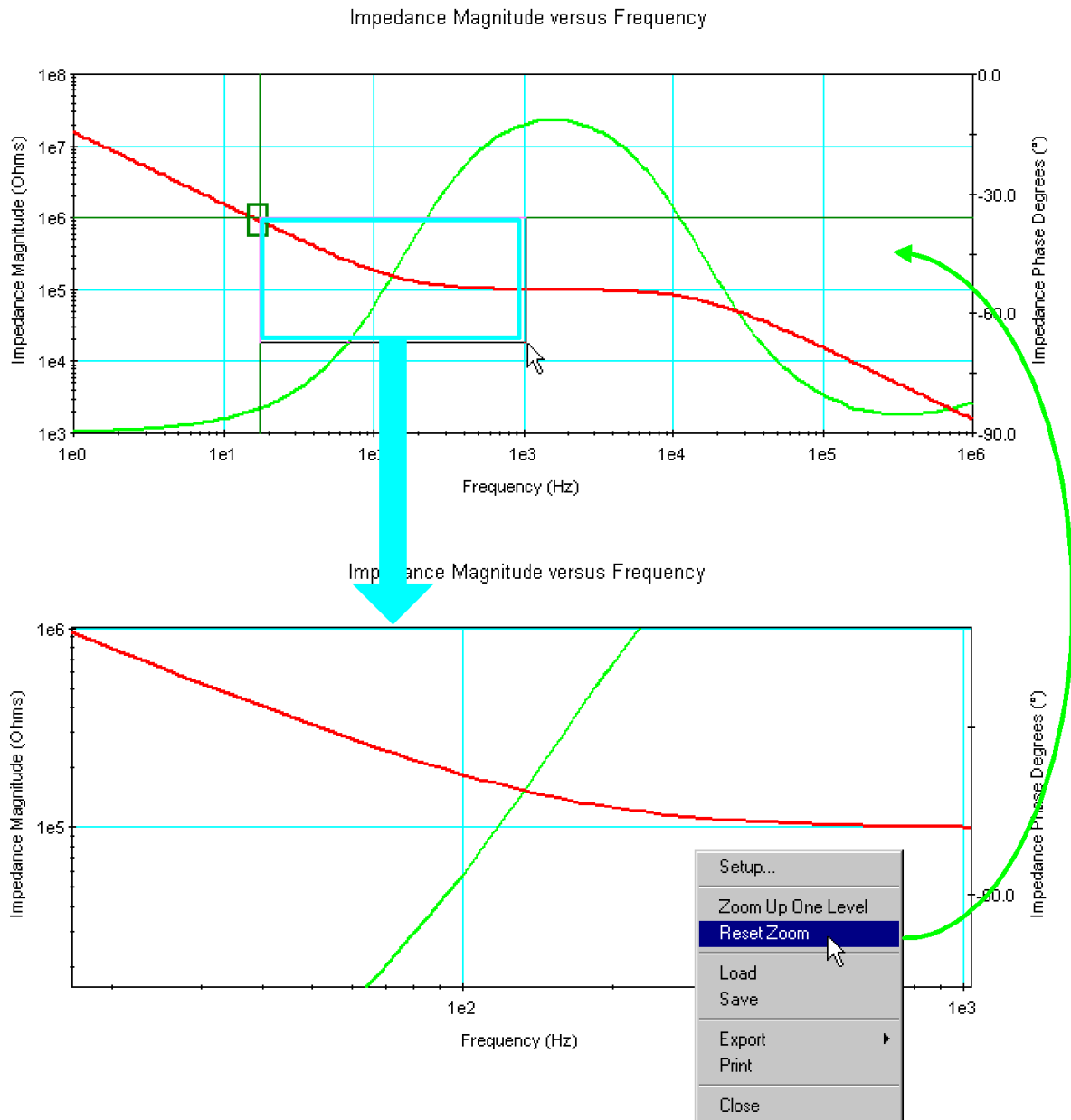


Figure 7-7 Example of graph zooming.

7.3 TABULATING YOUR RESULTS

Selecting **View Data** on the **Data File Information** window displays your results in tabular form:

The screenshot shows a window titled "Results Data - 'D:\Program Files\SMART\Project 1\Exp1.dat'". The window contains a table with the following data:

Sweep No.	Point No.	Time	Frequency (Hz)	AC Level	DC Level	Temperature	Impedance Real (Ohms)	Impedance Imag (Ohms)	Impedance Mag (Ohms)
1	1	00:01:37	10.000Hz	1.000V	0.000V	0.000K	99.507	-249.592m	99.507
1	2	00:03:10	21.544Hz	1.000V	0.000V	0.000K	99.506	-547.945m	99.507
1	3	00:04:42	46.416Hz	1.000V	0.000V	0.000K	99.550	-1.243	99.558
1	4	00:06:14	100.000Hz	1.000V	0.000V	0.000K	99.522	-2.843	99.562
1	5	00:07:46	215.443Hz	1.000V	0.000V	0.000K	99.380	-6.292	99.579
1	6	00:09:20	464.159Hz	1.000V	0.000V	0.000K	97.559	-13.543	98.495
1	7	00:10:38	1.000kHz	1.000V	0.000V	0.000K	91.243	-27.219	95.216
1	8	00:11:50	2.154kHz	1.000V	0.000V	0.000K	70.327	-44.928	83.453
1	9	00:12:46	4.642kHz	1.000V	0.000V	0.000K	34.442	-46.856	58.152
1	10	00:13:31	10.000kHz	1.000V	0.000V	0.000K	10.483	-29.881	31.666

On the right side of the window, there are three buttons: "Previous Block" (up arrow), "Next Block" (down arrow), and "Close" (red X). At the bottom right, there is a magnifying glass icon over a grid.

If you do not select a particular unit and channel with **Units/Channels used...** then CellTest accesses all the data in the file.

2. Data are stored in blocks of 1000 results. To help you search through a large amount of data, you can **go to** the start of a data block specified by the **Start at (x1000)** slider bar. Thus, you can start at result 0, 1000, 2000, and so on. As an additional aid, you can step through your selected block of 1000 results in smaller blocks, whose size you specify with the **Block size** slider bar. Use the **Next Block** and **Previous Block** buttons for block selection.

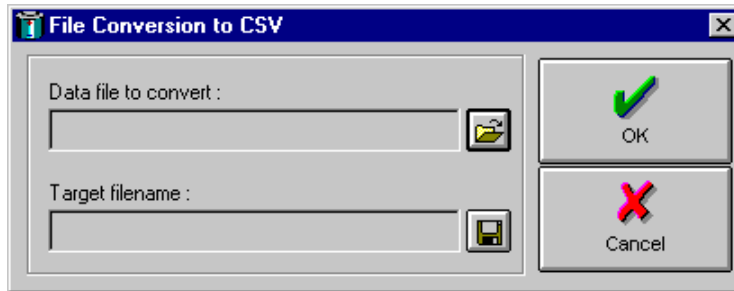
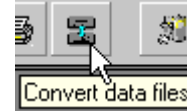
You can also display your results in an Excel table, which enables you to plot the results data in various ways. Before you can do this, however, you must first convert the data file to CSV format. See Section 7.4.

7.4 CONVERTING THE DATA FILE FORMAT

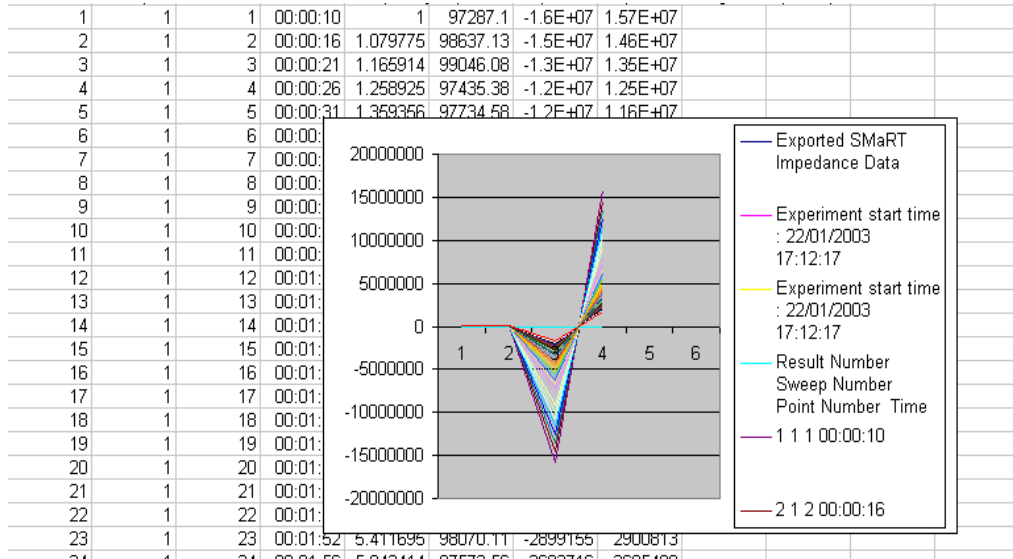
Should you wish to export a data file to other applications, you must first convert it into an appropriate format. CellTest converts the data files to .CSV format, which is suitable for the export of tabular data to Excel.

The conversion procedure is:

1. Click on the **Convert** button on the **Data File Information** window or on the **Convert data files** icon in the toolbar or, in the **Tools** menu, click on **File Conversion...** This reveals the **Data File Conversion** window:



2. Select the **Data file to convert.**
3. Select the **Target filename.**
6. Click on **OK**. Your selected data file is now converted and stored under the target filename. The converted file is displayed in Excel. The graph facility in Excel allows you to display the data in many different formats.



7.5 NORMALISING THE MEASUREMENT DATA

Normalisation is used primarily to cancel out errors in difficult measurement environments. A typical application in which normalisation can be usefully applied is where dielectric tests are done on samples that are heated or cooled by a temperature controlled cryostat system. The long wires with which the sample is connected are subject to many sources of stray impedance. It is impractical to use the reference measurement mode, since this would require a second cryostat of exactly the same construction as the one containing the sample. Here, measurement normalisation can be used to full advantage.

Measurement normalisation corrects for the effect of stray impedances due to such things as the connection cables, sample holder, and temperature control system. The measurements are made as quickly as possible, since only the sample is measured –not the reference capacitor, as is the case in the reference measurement mode.

Normalisation is applied at all the measurement frequencies of interest. A frequency sweep is made with the cryostat, sample holder and connecting leads set up as for the experiment, but without the sample inserted. This means that the sample holder is measured as a capacitor with an air dielectric. Each measurement result thus obtained comprises the impedance of the empty sample holder, plus any stray impedance in the measurement system. From the dimensions you enter as the **Sample Definition**, SMaRT calculates the impedance of the empty sample holder at each measurement frequency. This gives a correction factor for each measurement frequency, which you save in an empty call data file for later use.

Note that normalisation is most effective when the measured impedances are similar to the calculated values used in normalisation. This can sometimes be arranged by varying the separation of the sample holder plates.

The normalisation file for a particular experiment is created with a single sweep. The sample may then be put into the sample holder, normalisation requested from the specified normalisation file, and measurements made at different temperatures.

You can use measurement normalisation to advantage whenever you wish to capture data quickly and accurately. Typical applications are in the measurement of fast changing systems, such as the cure of adhesives and thermo set polymers, etc.

7.5.1 NORMALISATION PROCEDURE

The procedure for making a normalised measurement is:

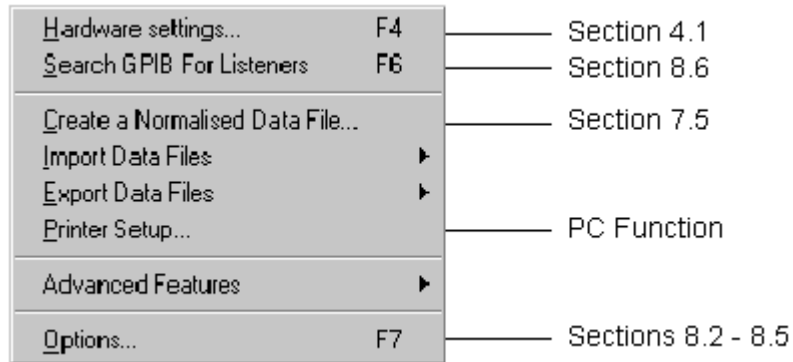
1. Mount the sample in the sample holder and connect the sample holder to the **Sample** inputs of the 1296.
2. In the **Experiment Editor**, enter the experiment set-up, including the sweep and measurement parameters and the name of the **Data file** that is to hold the measured sample data.
3. **Save** the experiment and **close** the **Experiment Editor**.
4. Run the experiment, by clicking on the traffic light icon, choosing the experiment set up in Step 2, and selecting **Start**. On the successful completion of this experiment, an icon representing the measured sample data file appears on the **Project Viewer**.
5. Double-click on the sample data file icon to reveal the **Data File Information** window and use this to ascertain the sample capacitance.
6. Remove the sample from the sample holder.
7. Edit the experiment set up in Steps 2 and 3 to specify the **Data file** that is to hold the empty cell data.
8. In the **Experiment Editor**, enter the **Sample Definition** –that is, the dimensions of the empty sample holder. Specify the Sample Thickness to obtain an Empty Cell Capacitance that is approximately the same as the sample capacitance value obtained in Step 5.
9. **Save** the experiment and **close** the **Experiment Editor**.
10. Set the electrode gap of the sample holder to the same value as the sample thickness specified in Step 8.
11. Re-run the experiment, by clicking on the traffic light icon, choosing the experiment edited in Steps 7 and 8, and selecting **Start**. On the successful completion of this experiment, an icon representing the empty cell data file appears on the **Project Viewer**.
12. From the Tools menu, select '**Create a Normalised Data File**'. A **Normalisation** window appears.
13. In the **Normalisation** window, enter the names of the measured sample data file and the measured empty cell data file. A default name is presented for the file that is to hold the normalised data, but you can change this if you wish. Then select **OK** to create normalised data file. The **Data File Information** window now appears, which enables you to select and display your normalised data.

8. Support Features

8.1 USING THE SUPPORT FEATURES

The SMaRT support features are available in the **Tools** menu.

On the top level of this menu are the followings items:



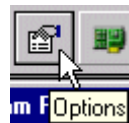
Some options are described elsewhere in the User Guide. The **Add/Remove Hardware** option is in Section 4.1 and the **Normalised Data File** option is in Section 7.5.

The Advanced Features are diagnostic tools. These should be used only under the direction of a qualified engineer or service technician, should the need arise.

Printer Set-up is a standard Windows function and its use is explained in the Microsoft help text.

The remaining features are described in the present section, as shown above.

Note that you can also access the options described in Sections 8.2 through 8.5 by clicking on the **Options** icon in the toolbar.



8.2 SELECTING THE GENERAL OPTIONS

Figure 8.1 shows the Cell Test **General** options (default settings).

The use of each item in this window is self-explanatory, if you are familiar with the terms used in cell testing with the 1294 or 1296. If you are not, then read Section 1 of the User Guide.

Some options are selected from list of settings, others by ticking a box.

To disable an option, click to remove the tick in the adjacent box. Alternatively, to reinstate the option, click to tick the box.

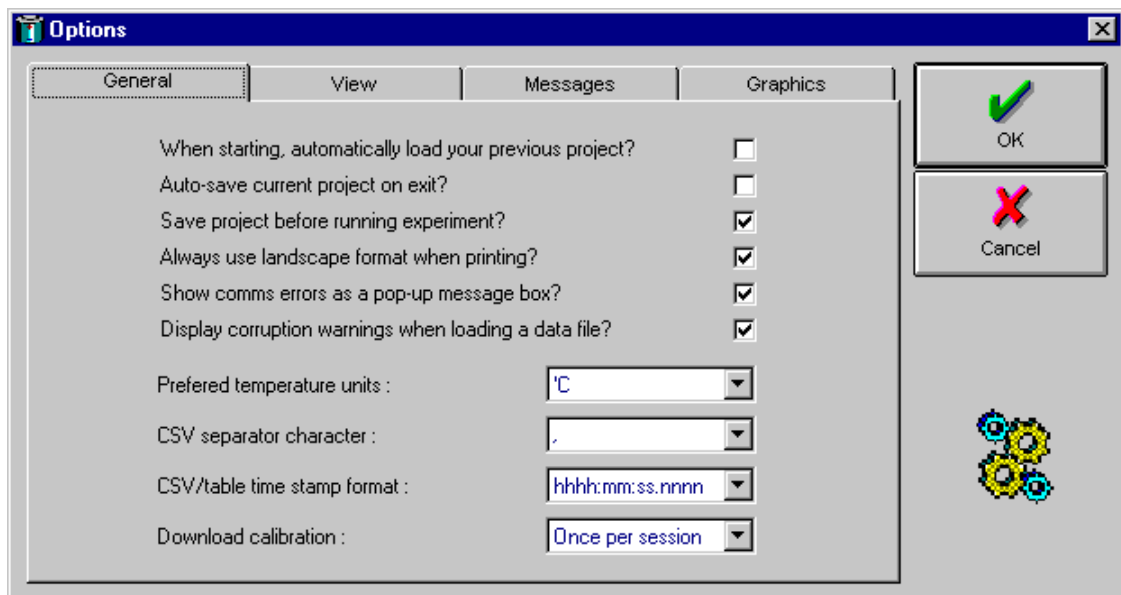


Figure 8-1 General options window.

8.3 SELECTING THE VIEW OPTIONS

Figure 8.2 shows the **View** options window. The significance of most settings is self-explanatory. Examples of the various project tree styles are shown in Figure 8.3.

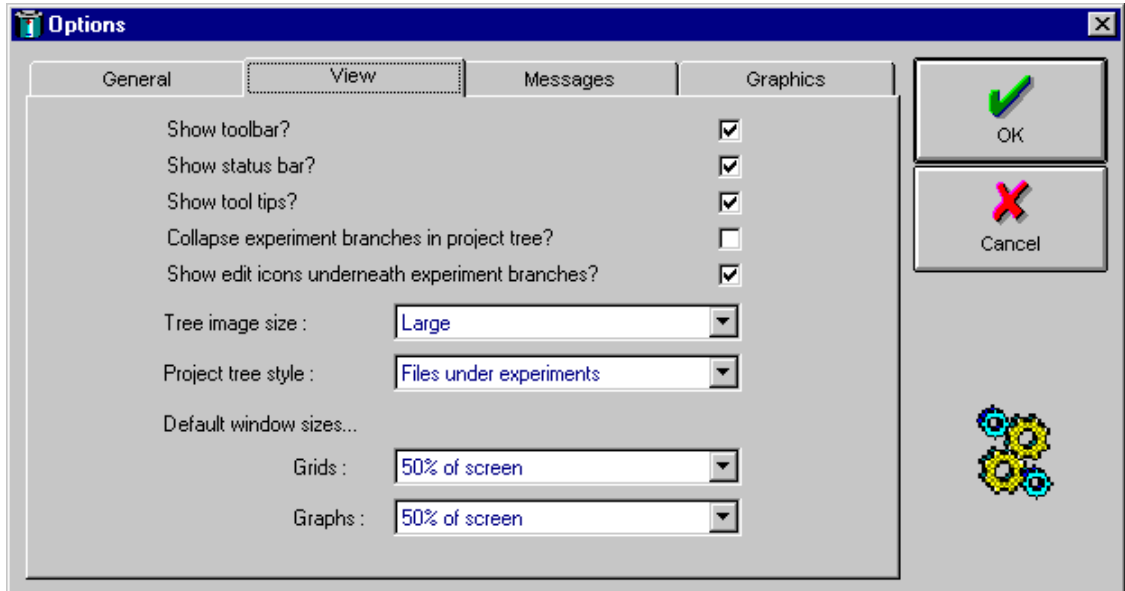


Figure 8-2 View option window.

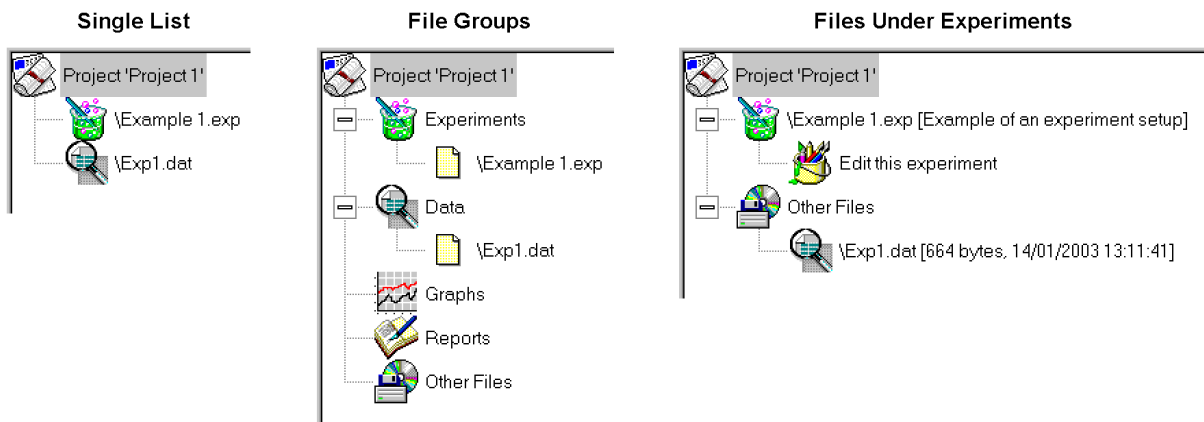


Figure 8-3 Project tree styles.

8.4 SELECTING THE MESSAGES OPTIONS

Figure 8.4 shows the message options (default settings).

The use of each item in this window is self-explanatory, if you are familiar with the terms used in cell testing with the 1294 or 1296. If you are new to these, then read Section 1 of the User Guide.

To disable an option, click to remove the tick. To reinstate, click to tick.

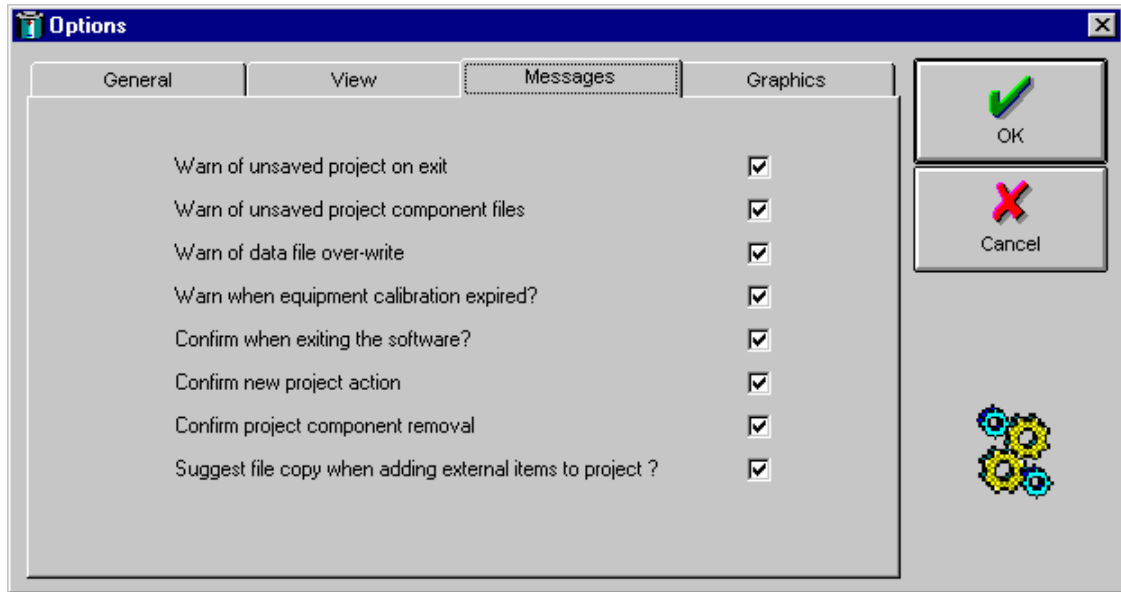


Figure 8-4 Messages options window.

8.5 SELECTING THE GRAPHICS OPTIONS

Figure 8.6 shows the graphics options (default settings). The use of each item in the window is mainly self-explanatory.

The third option down means 'Do you want to be asked whether you want to change the graph set-up, when the experiment starts?'

To disable an option, click to remove the tick. To reinstate, click to tick.

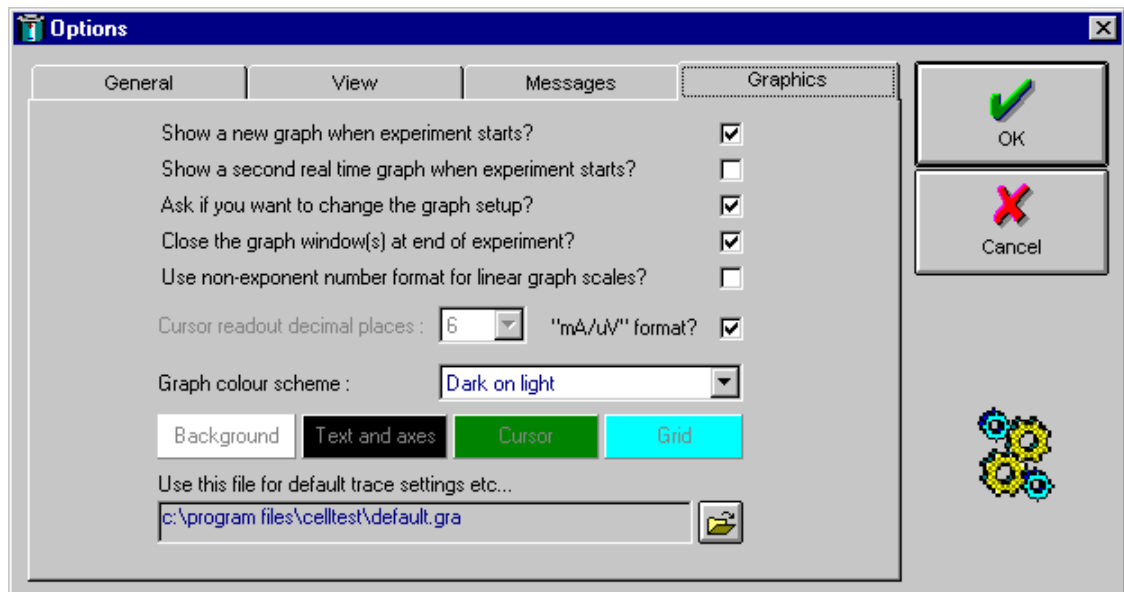



Figure 8-5 Graphics option window.

8.6 SEARCHING THE GPIB FOR LISTENERS

Searching the GPIB for listeners allows you to check which instruments in your System are operational and available for use.

To start a search, click on the  icon in the toolbar or select **Tools** → **Search for GPIB Listeners**. At the end of the search, a list of available instruments is displayed.

9. Index

- Analyser integration, selecting the.. 5-8
- Analyser ranging and coupling (1294)
 - selecting the 5-10
- Converting the data file format 7-10
- Creating a Project
 - basic procedure 4-2
 - project files..... 4-1
- Creating an experiment 1-2, 5-3
- Editing an experiment 5-4
- Experiment editor, using the 5-1
- Generator balancing (1294) selecting the
 - 5-11
- Managing a Project 4-4
 - adding files from other projects.... 4-5
 - loading a project..... 4-4
 - saving a project 4-5
- Measurement method for 1296,
 - selecting the 5-9
- Normalising measurement data 7-11
 - procedure 7-12
- Plot traces, setting up the 1-3, 7-3
- Plot type, selecting the..... 7-2
- Plot type, selecting the..... 1-3
- Plot view, setting up the 7-6
- Plot x-axis, setting up the 1-4, 7-4
- Plot y-axes, setting up the 7-5
- Plot y-axis, setting up the 1-4
- Project files..... 1-2
- Project, creating a 2-3
- Reference capacitor (1296), external,
 - selecting the..... 5-9
- Set-up entries 1-1
- SMaRT, introduction to use of..... 2-1
- Starting an experiment..... 6-1
- Stopping an experiment..... 6-2
- Support features, using the 8-1
 - general options..... 8-2
 - graphics options..... 8-5
 - messages options 8-4
 - search GPIB for listeners 8-5
 - view options 8-3
- System hardware, entering the set-ups
 - 3-1
- Tabulating your results 7-1, 7-9
- Viewing your results 1-4
- Zooming in on the graph detail..... 7-8

