

METRICS WIN4145

REFERENCE GUIDE

CONTENTS

CHAPTER 1: A QUICK TOUR OF WIN4145

[How Win4145 Stores Information](#)

[Project Files](#)

[The Win4145 Initialization File](#)

[Data and Plot Windows](#)

[The Menu Bars](#)

[The Toolbar and Status bar](#)

[Engineering Units](#)

CHAPTER 2: THE SETUP EDITOR

[Information Stored in a Test Setup](#)

[The Test Setup/Project File Relationship](#)

[Test Setup Applications](#)

[The Setup Editor](#)

[Creating a Single Test Setup](#)

[Creating a Time Domain Test Setup](#)

[Creating Multiple test Setups](#)

[Setup Editor Controls](#)

CHAPTER 3: THE FILE MANAGER

[What is a Project File](#)

[Opening a Project File](#)

[Saving a Project File](#)

[Deleting Project Files](#)

CHAPTER 4: MEASURING AND TESTING DEVICES

[Measurement Types](#)

[Measurement Modes](#)

[Interactive Measurement Controls](#)

[Executing a Test Setup](#)

[Executing a Sequence of Test Setups](#)

CHAPTER 5: DATA WINDOWS AND DATA ANALYSIS

[What is a Data Window](#)

[Exploring the Data Menu Bar](#)

[Creating Data Window Spreadsheets](#)

[Editing Data Window Spreadsheets](#)

[Auto Analysis with the Transform Editor](#)

[Transform Editor Functions by Category](#)

[Transform Editor Function Definitions](#)

CHAPTER 6: PLOT WINDOWS

[What is a Plot Window](#)

[The Plot Menu Bar](#)

[Plot Window Tools](#)

[Creating a Plot Window](#)

[Editing a Plot Window Definition](#)

[Building Data Groups for Plot Displaying](#)

[The Plot Window Tool Ribbon](#)

[The Curve Fit Tool Set](#)

[The Options Tool Set](#)

CHAPTER 7: PRINTING, EXPORTING, AND IMPORTING

[Printing and Plot Window Contents](#)

[Copying a Data or Plot Window to the Clipboard](#)

[Sharing Data Between Win4145 Workstations](#)

[Extracting ASCII Data from Win4145](#)

CHAPTER 1: A QUICK TOUR OF WIN4145

WHAT IS WIN4145 SOFTWARE?

Win4145 is a uniquely powerful instrumentation control and data analysis software package. Win4145 is designed to control the Hewlett Packard 4145A and 4145B semiconductor parameter analyzer used for device characterization and other microelectronics testing.

- Win4145 operates in the Microsoft Windows NT 4.0, 2000, or XP Professional environment.
- Win4145 eliminates the need for any programming or front-panel interaction with the HP 4145.
- The HP 4145 is controlled from a user-interface configured by point-and-click editing.
- Win4145 delivers more than programming-free instrumentation control.
- Win4145's spreadsheet windows and scientific plotting capability allow you to view and analyze data easily.
- Win4145 also includes valuable scientific and engineering data reduction tools necessary for interpreting test results, such as cursor assignments and curve fitting routines.
- Win4145's resources also include a robust file management system.
- Win4145's file management system allows the user to create multiple databases without relying on external database software.

HOW WIN4145 STORES INFORMATION

All of the information that is either specified or measured in Win4145 is stored in one of three possible locations:

1. A Test Setup
2. A Project File
3. The ICS35.INI File

Once all the necessary specifications are configured for a given hardware arrangement, these specifications never have to be configured again unless you change your existing hardware arrangement. This means that all you have to do to re-measure a test setup from a cold start of Win4145 is

1. Select the appropriate Project File
2. Select the desired Test Setup
3. Execute the measurement

TEST SETUPS

A test setup stores the instrument setup configurations required for a particular measurement. If the Setup Editor is used as recommended, the test setup will also document the hardware connections between the instrument and the device required for the associated measurement. Examples of some test setups include the collector characteristics of a bipolar transistor, collector breakdown voltage, gate threshold voltage, etc.

INFORMATION STORED IN A TEST SETUP

A test setup includes at least three pieces of information, and in some cases four:

1. Win4145 provides the capability to configure the setup of each instrument source unit in a graphics-oriented workspace. This workspace eliminates front-panel interaction with the instrument and eliminates the need for any programming. Select the appropriate Project File
2. Win4145 provides the capability to configure the setup of each instrument source unit in a graphics-oriented workspace. This workspace eliminates front-panel interaction with the instrument and eliminates the need for any programming.
3. The Setup Editor workspace is provided as a tool to document the hardware connections or device orientation required for the corresponding device measurement. While it is recommended that the connections designated in the Setup Editor correspond to the reality of your hardware arrangement, it is not necessary. If the required instrument/DUT connections are accurately designated in the Setup Editor, make certain that the instrument and the device are physically connected as designated before executing the test setup.
4. A source unit conventionally sources voltage and measures current, or sources current and measures voltage. Win4145 provides the capability to return both voltage and current regardless of which quantity is forced. When a test setup combines *swept* sources with *stepped* sources, Win4145 can record an entire family of data vector curves.

For example, a source unit may apply a voltage sweep across the drain of a MOSFET and return both the voltage sweep and the measured current, while a different source unit applies a voltage bias to the gate. After the first source unit sweeps the drain, the source unit at the gate will step to a new bias voltage in preparation for a second drain sweep. The result of this test setup is a family of drain current data vectors labeled I_{ds1} , I_{ds2} , I_{ds3} , etc. Each data vector corresponds to a single curve.

Win4145 allows you to specify what data each source unit will return, and it allows you to specify a root label for each family of returned data (sequence numbers are automatically appended by Win4145).

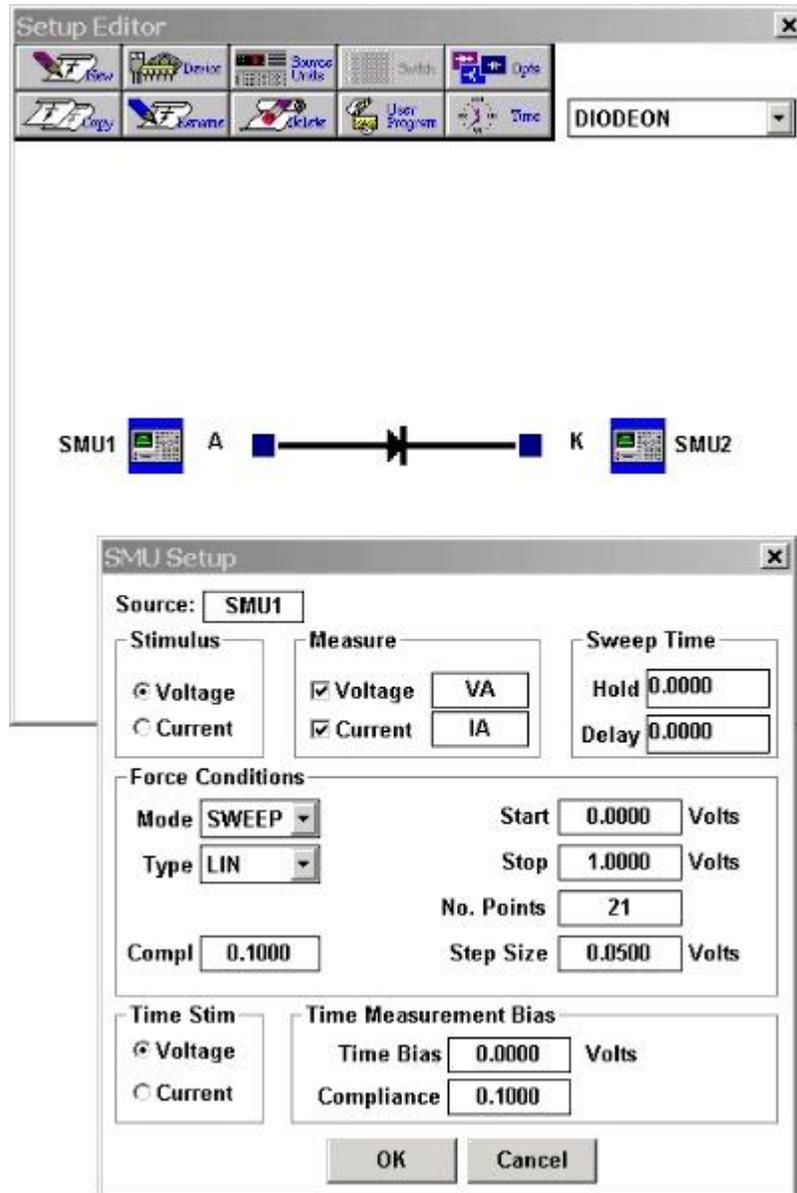


Figure 1-2: The test setup stores instrument setup conditions, documents the instrument/DUT connections, stores what data is returned and how it is labeled.

THE TEST SETUP/PROJECT FILE RELATIONSHIP

A test setup defines the instrument configurations for a specific device measurement. Test setups are created in the Setup Editor. A project file stores one or more test setups as a single unit. Project files are defined in the File Manager. A project file will normally contain all of the test setups corresponding to a single device or wafer. All of the data associated with a test setup is written to a single data window spreadsheet. Each data window spreadsheet corresponds to a single test setup.



It is not recommend that the user create more than fifteen test setups in a single project file in Win4145. If this limit is exceeded the stability of the operating system may deteriorate.

Test setups are only accessible from the project file (or temp file) in which they were created. A test setup cannot be copied and moved to another project file. The exception to this rule occurs when you use the Auto-store feature, but this is a higher function of Win4145 covered elsewhere in this manual. In general, the latter point is true.



A test setup cannot be copied to another project file.



Figure 1-3: Test Setups corresponding to a Project File are displayed along the bottom of the Desktop.

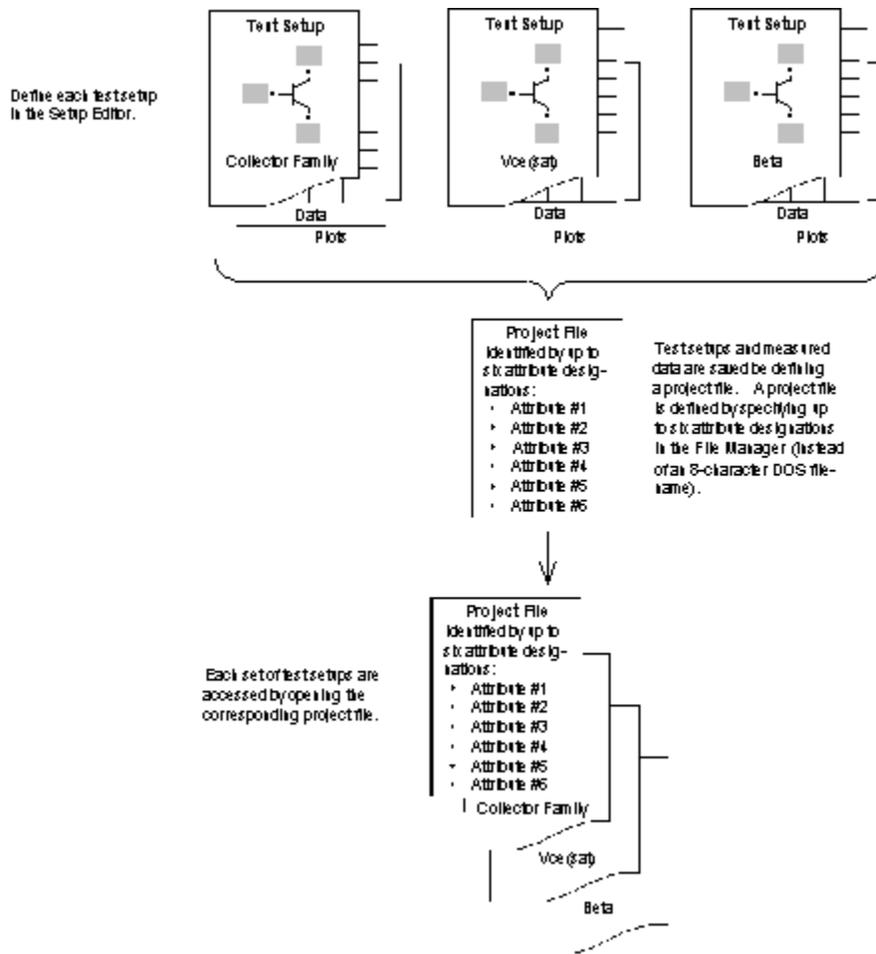


Figure 1-4: One or More Test Setups make up a Single Project File. A Test Setup is only Accessible from the Project File or Temp File in which it was created.

TEST SETUP APPLICATIONS

Consider the need to characterize the performance of a device under different electrical and environmental conditions. You may create a group of test setups in which each test setup represents a different set of characteristic curves. For example, one test setup may measure drain family characteristics, another gate transfer, and still another sub-threshold drain characteristics.

THE SETUP EDITOR

Each test setup is uniquely created, named, edited, or deleted with the use of the Setup Editor. The Setup Editor is a graphic environment that is completely icon and menu-driven.

THROUGH POINT-AND-CLICK CONTROL, THE SETUP EDITOR PROVIDES THE CAPABILITY TO:

1. Specify the instrument setup configurations of each source unit.
2. Document the connections between the instrument and device.
3. Specify what data is returned and how it is labeled.

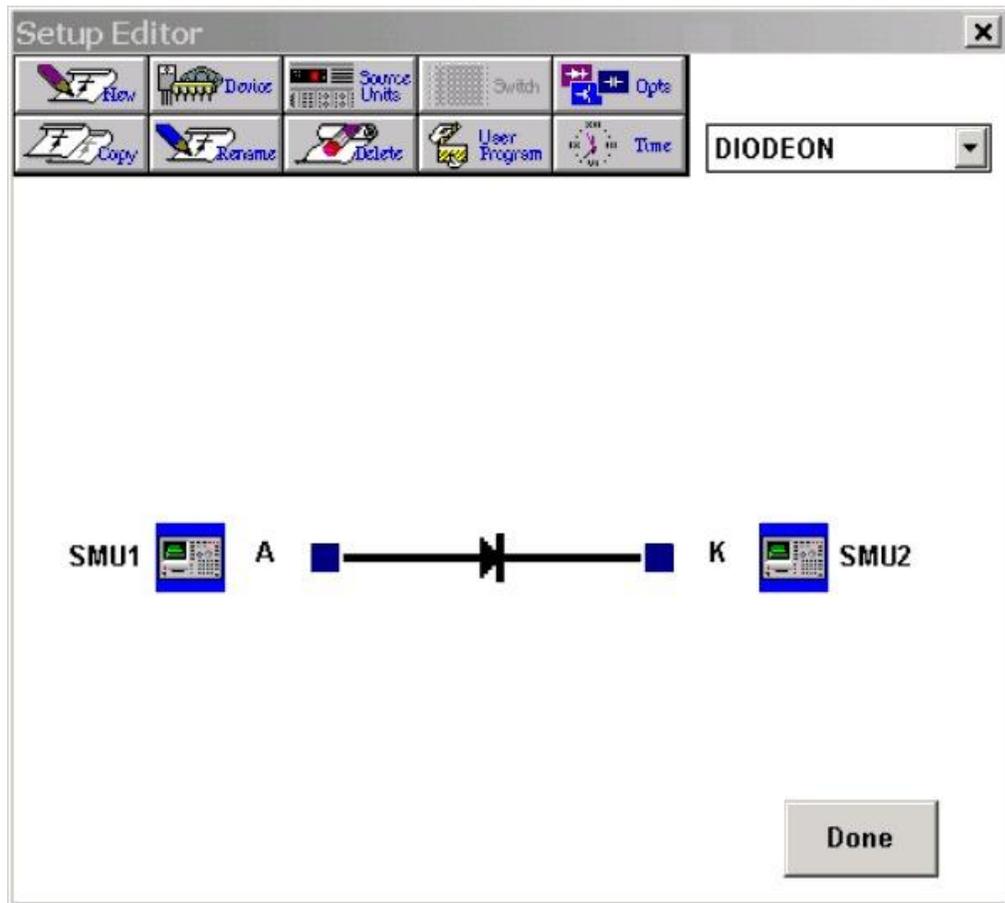


Figure 1-5: The Setup Editor is Used to Create Test Setups

PROJECT FILES

A project file is a collection of one or more test setups stored as a single unit. A project file provides a second level of organization allowing you to store a group of test setups that share a common association. By creating a project file, you can manipulate a group of related test setups as a single entity. A project file can be individually saved, deleted, imported, or exported.

A PROJECT FILE STORES TWO CATEGORIES OF INFORMATION

1. A project file stores the associated test setup(s).
2. A project file stores any of the measured data and plots corresponding to the test setups.



When a project file is opened, any of the defined test setups can be executed immediately.

CREATING PROJECT FILES



A project file is opened by displaying the File Manager and selecting the directory and filename designations specified when the project file was saved.

Selecting OK in the File Manager will open the project file and display the corresponding test setups at the bottom of the desktop.

For more information about configuring the project file storage mode or customizing the File Manager attribute labels, please refer to Chapter 3: *Setting the File Manager Options*.

PROJECT FILE APPLICATIONS



A project file allows you to save a related group of test setups and measurement results as a single entity.

For example, you can create a project file in which each test setup measures a unique device characteristic on the same device, or you can create a project file in which each test setup measures the same device characteristic at different wafer or test fixture locations. Alternatively, you can create a project file that includes test setups of both types.

MULTIPLE MEASUREMENTS/SINGLE DEVICE

Consider an application in which you are testing four bipolar transistors at three different temperatures. We'll assume your testing requirements include a family of collector curves, a forward gummel plot, and a VCE(sat) curve. Each of the three characteristics just mentioned will comprise three separate test setups. However, the results of a single device measured at a single temperature will comprise a single project file. The final result of this example will include twelve project files (4 devices x 3 temperatures). Each project file will include the three test setups described above. In this example the project file storage mode will be set to "Package". This will reduce the number of available File Manager attribute fields from six to three. A schematic of the project file designations is shown on the following page.

After the data is returned, click the SAVE AS toolbar button and specify a filename #1 designation, such as "2N2222 BJT 32C". Click the File Manager OK button to accept the project file designations and save your results.

Expose the device to the second temperature and re-execute the three test setups or test setup sequence. The most recently measured data will overwrite the current data and corresponding plot windows without overwriting the data just saved as a project file (as long as the auto-store function has not been activated in the Output Data control). Click the SAVE AS toolbar button and update the temperature specification in the File Manager filename #2 with designation 2N2222 BJT 125C. Click the File Manager OK button to accept the project file designations and save your results. You have just created a second project file.

Continue changing your device and temperature as necessary being sure to select the SAVE AS function after the corresponding measurements. The SAVE AS function will create a new project file. If you select the SAVE function you will overwrite the previously defined project file with the current results.

THE WIN4145 INITIALIZATION FILE (ICS35.INI)

The ICS35.INI file is stored in the Win4145 directory by default. The ICS35.INI file is created during the installation of the Win4145 Program. The file is updated automatically as a result of certain procedures executed within Win4145.

The ICS35.INI file is read by Win4145 each time Win4145 is started or a driver is connected. The settings defined in the ICS35.INI file DO NOT have to be specified each time Win4145 is started.

THE ICS35.INI FILE WILL BE UPDATED WHEN

1. The communications hardware installed in your computer changes.
2. The communications address of the 4145 instrument changes.
3. Win4145 is customized by changing the workspace controls.

THE ICS35.INI FILE IS DIVIDED INTO FOUR SECTIONS

1. Environment Specifications - The specifications included in this section define the locations of the Win4145 directories and sub-directories. These specifications are defined by the user during the installation of the Win4145 Program.
2. Communications Interface Specifications - These specifications identify the communications type installed on your computer. This section also includes a few functional specifications necessary for the proper operation of the communications interface. These specifications are initially configured during the installation of the Win4145 Program, but they can be reconfigured while running Win4145. If you replace the communications card installed in your computer, you can update the ICS35.INI file without the need to reinstall Win4145. Refer to the Getting Started Guide, *Designating the Communications Interface Board*
3. Instrument Designations - Once specified, GPIB addresses are written to the ICS35.INI file. Each time the instrument driver is connected, the corresponding GPIB address is set automatically.
4. Workspace Controls - The workspace controls are those specifications that define the customized features of Win4145, such as the File Manager Setup options or the default plot colors. These specifications are configured in the Workspace dialogue box opened by selecting OPTIONS/WORKSPACE from the Win4145 menu bar.

DATA AND PLOT WINDOWS

As soon as a test setup is defined, a corresponding data window spreadsheet will be created and minimized at the bottom of the Win4145 desktop. Win4145 will identify the corresponding data window spreadsheet with the same name as the test setup; however, the data window spreadsheet will be empty prior to executing any measurement.

The example presented the following page shows a data window for the test setup entitled "COLLECTOR FAMILY". The minimized windows arranged along the bottom of the desktop are additional data window spreadsheets that belong to the project file.



Each data window spreadsheet corresponds to a single test setup, and only one test setup can correspond to a single data window spreadsheet.

The dark icons at the bottom of the desktop are minimized plot windows that correspond to particular data window spreadsheets.

	VA	IA	VON +
1	0.0000	-2.3000p	0.6218
2	0.0500	0.7895n	
3	0.1000	3.8235n	
4	0.1500	0.0136u	
5	0.2000	0.0429u	
6	0.2500	0.1312u	
7	0.3000	0.4074u	
8	0.3500	1.3165u	
9	0.4000	4.3870u	
10	0.4500	0.0150m	
11	0.5000	0.0512m	
12	0.5500	0.1705m	
13	0.6000	0.5417m	
14	0.6500	1.5925m	

Figure 1-10: An Example of a Win4145 Data Window Spreadsheet.



Plot windows are created by the user from the measured or calculated values contained in a corresponding data window spreadsheet. Up to ten plot windows can be associated with a single data window.



Each plot window is dynamically linked to its corresponding data window. This means that once a plot window is created, the contents of the plot window will be updated each time the test setup is executed.

The plot window can also be updated upon user command each time the data window is edited. Similarly, the contents of a data window will be updated each time the associated test setup is executed.

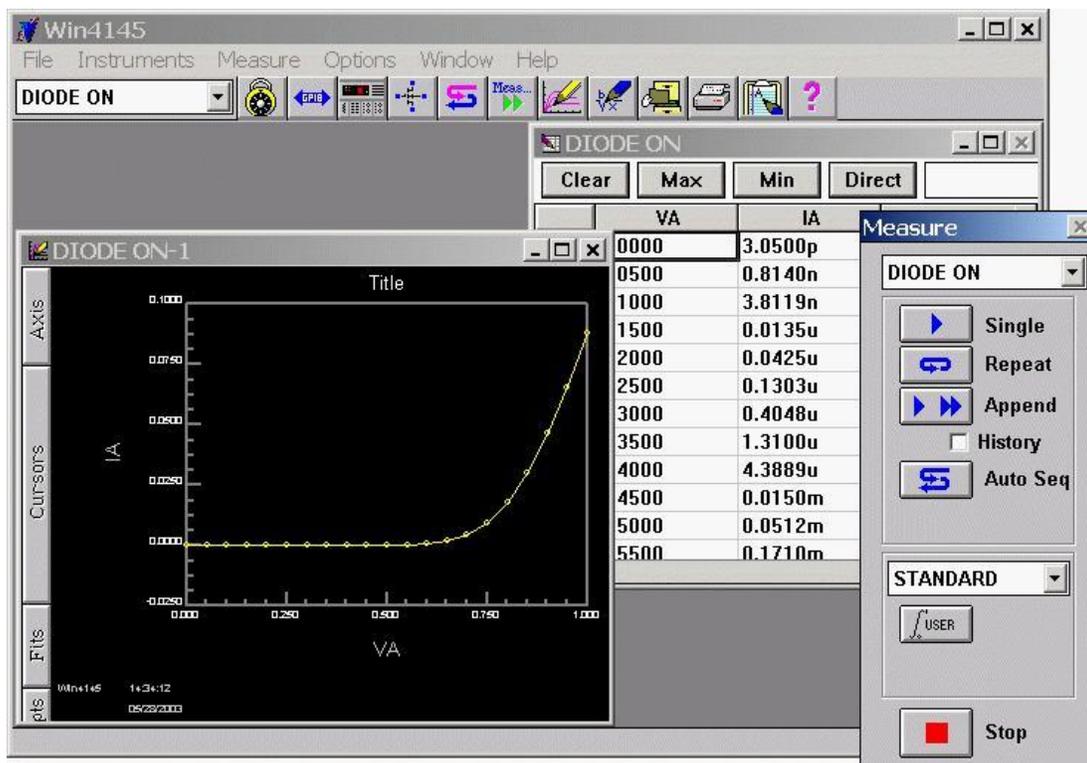


Figure 1-11: A Plot Window is Dynamically Linked to the Corresponding Data Window Spreadsheet.

THE MENU BARS

Win4145 operates in one of three functional modes: the Measurement Mode, the Data Mode, and the Plot Mode. The primary difference between each functional mode is the group of activities that each mode supports. The visual distinction between each functional mode is the choice of menu options available in each mode's menu bar. These menu choices provide different capabilities for different phases of device testing. Each functional mode will present slightly different menu choices; however, all three modes include the File, Window, and Help pull-down menus.

MEASUREMENT MODE

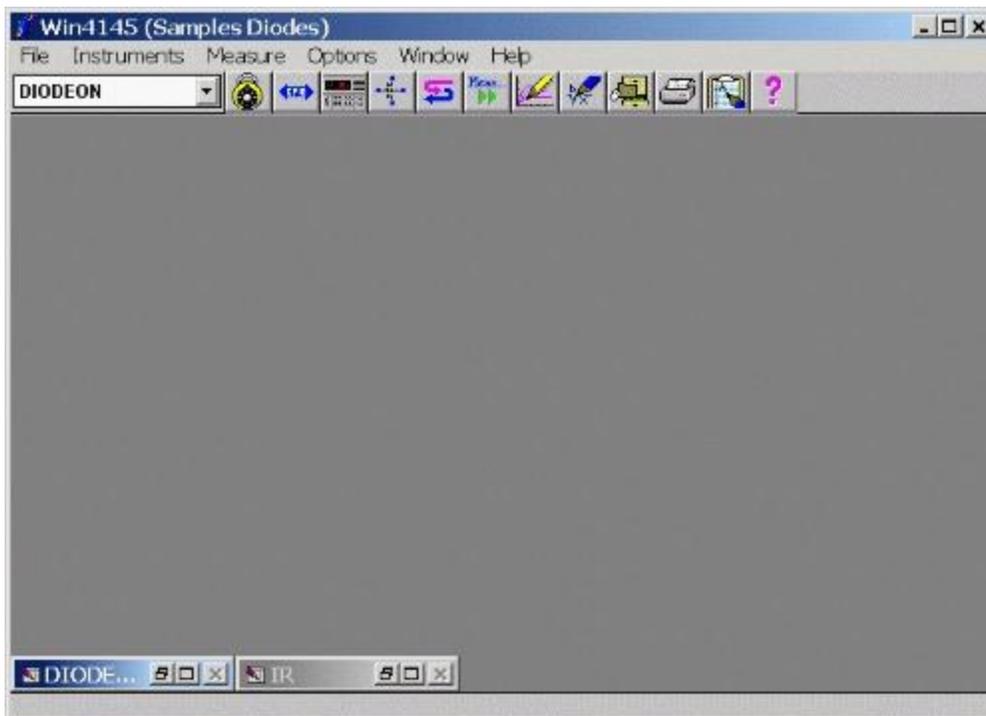


Figure 1-12: The Measurement Mode Menu Bar.

When Win4145 is started, Win4145 defaults to the measurement mode. The measurement mode is the mode from which test setups are executed. The measurement mode is also the mode from which all of the test options are defined. Defining GPIB addresses, selecting instrument drivers, and creating test setups are all completed while in the measurement mode.



Regardless of what menu bar is active, the measurement mode menu bar can always be activated by clicking on the right mouse button. If data or plot windows are active, the measurement mode menu bar can also be accessed by clicking the mouse anywhere in the desktop.

DATA MODE

The data mode menu bar is displayed when a data window is active. The drop-down menus that are unique to the data mode menu bar include the Edit, Options, and Analysis menus. The data mode can be activated by clicking on any data window.



Figure 1-13: The Data Mode Menu Bar.

PLOT MODE

The plot mode menu bar is displayed when a plot window is active. The drop-down menus that are unique to the plot mode menu bar are the Setup, Options, and Analysis menus. The plot mode Options and Analysis menus do not contain the same choices as the Options and Analysis selections in the data mode. The plot mode can be activated by clicking on any plot window. An example of the plot mode menu bar is shown on the following page.



Figure 1-14: The Plot Mode Menu Bar.

THE TOOLBAR AND STATUS BAR

Win4145 is shipped with the toolbar and status bar displays defaulted to ON. Both the toolbar and status bar can be independently toggled between ON/OFF. The toolbar and status bar displays can be toggled for the current execution of Win4145. The *default* display configurations can also be changed in the Workspace Controls dialogue box.

HOW TO TOGGLE THE TOOLBAR AND STATUS BAR DISPLAYS DURING THE CURRENT EXECUTION OF WIN4145

1. From the measurement mode menu bar, select OPTIONS/TOOLBAR, or OPTIONS/STATUS BAR.
2. Selecting either the toolbar or status bar option will immediately toggle the respective feature to the opposite state.
3. The selection designated from the OPTIONS menu will remain active even if Win4145 is restarted.

THE TOOLBAR

The toolbar is designed to provide easy access to the features most likely needed while using Win4145. Each of the toolbar features are accessible through the appropriate drop-down list located in one of the menu bars, but the toolbar allows you to access the corresponding feature with a click of a single button.

The toolbar includes a setup window and twelve buttons. The functions of the setup window and each button are described below.

SETUP WINDOW

The setup window displays the active test setup. Click the corresponding scroll arrow to display a list of available test setups. The active test setup can be changed by selecting one of the available options.

CODEWORD SETUP



This button will open the Password Setup dialogue box. The Password Setup dialogue box is used to activate the Win4145 software. The codeword is required to activate the Win4145 software. Please refer to the Win4145 Installation Guide for instructions to set and reset your password.

COMMUNICATIONS SETUP



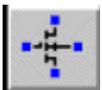
This button will open the Communications Setup dialogue box. The Communications Setup dialogue box is used to specify the communications type and associated parameters to Win4145.

SELECT INSTRUMENT



This button will open the Instruments dialogue box. The Instruments dialogue box is used to activate the HP4145 instrument driver within Win4145.

SETUP EDITOR



This button activates the Setup Editor. The Setup Editor will initially display the active test setup.

SEQUENCE EDITOR



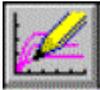
This button will open the Sequence Editor dialogue box. The Sequence Editor dialogue box is used to switch Win4145 between Single Test or Multiple Test mode as well as define a test setup sequence.

MEASURE



The measure button actuates the Measurement Remote Control dialogue box which is used to initiate all type of measurement.

CREATE PLOT



This button opens the Setup Data dialogue box. The Setup Data dialogue box is used to create a new plot.

TRANSFORM EDITOR



This button activates the Transform Editor. The Transform Editor is used to add new data vectors to a data window spreadsheet. These calculated data vectors are analytical expressions of measured data vectors.

SAVE PROJECT



This button opens the File Manager dialogue box allowing the user to define a new project file. A project file is defined by specifying up to six project file attributes.

PRINT



This button is used to send the contents of the active window to the printer.

COPY DATA TO CLIPBOARD



This button is used to send the contents of the active window to the clipboard.

ON-LINE HELP



This button is used to access the on-line help feature.

THE STATUS BAR

The status bar is located at the base of the Win4145 desktop. The status bar displays messages throughout the operation of Win4145. These messages either indicate what feature or function of Win4145 is currently active, or the messages prompt the user for an action.

ENGINEERING UNITS

Win4145 supports the use of engineering units in any of the numerical entry fields except the Transform Editor. For example, if the user designates "250u" in a numerical field, Win4145 will recognize this designation as 0.00025. Each engineering unit is a multiplier that acts on the numerical component of the field entry. When the user specifies "250u", Win4145 multiplies 250 by 10^{-6} to arrive at the numerical value of the field entry. A list of the engineering units recognized by Win4145 is summarized in the table below.

Notation	Multiplier
F	$\times 10^{-15}$
P	$\times 10^{-12}$
N	$\times 10^{-9}$
U	$\times 10^{-6}$
M	$\times 10^{-3}$
K	$\times 10^3$
M	$\times 10^6$
G	$\times 10^9$

Table 1-1: Engineering Notation Recognized by Win4145

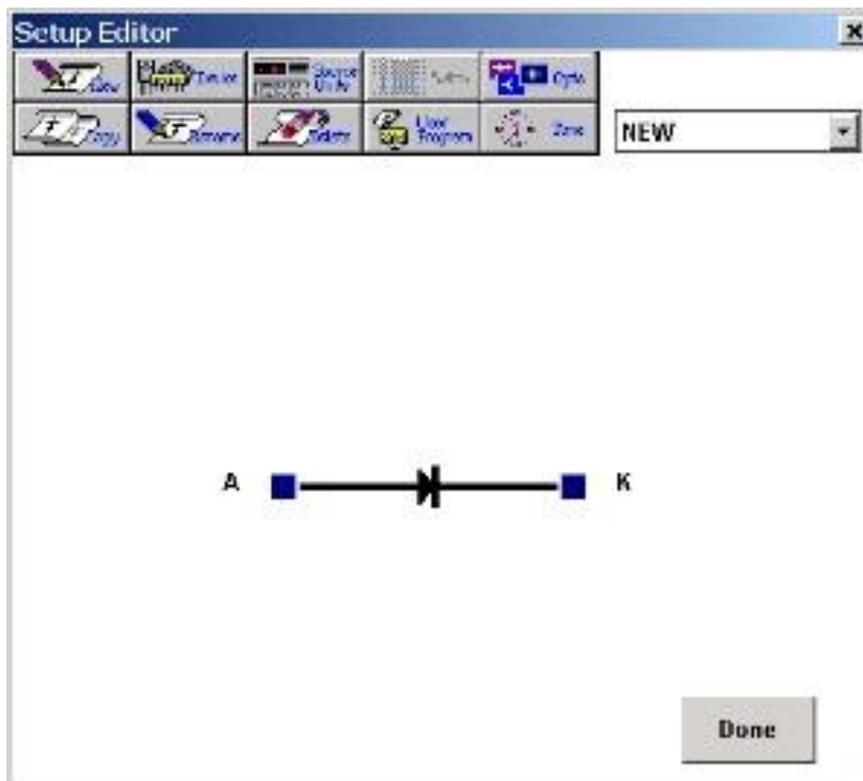
CHAPTER 2: THE SETUP EDITOR

WHAT IS A TEST SETUP?



A test setup stores the instrument setup configurations required for a particular measurement.

If the Setup Editor is used as recommended, the test setup will also document the hardware connections between the instrument and the device required for the associated measurement. Examples of some test setups include the collector characteristics of a bipolar transistor, collector breakdown voltage, gate threshold voltage, etc.



- 1. A Test Setup Defines the Instrument Setup Configuration of Each Source Unit:** - Win4145 provides the capability to configure the setup of each instrument source unit in a graphics-oriented workspace. This workspace eliminates front-panel interaction with the instrument and eliminates the need for any programming.
- 2. A Test Setup Documents the Connections Between the Instrument and DUT** - The Setup Editor workspace is provided as a tool to document the hardware connections or device orientation required for the corresponding device measurement. While it is recommended that the connections designated in the Setup Editor correspond to the reality of your hardware arrangement, it is not necessary. If the required instrument/DUT connections are accurately designated in the Setup Editor, make certain that the instrument and the device are physically connected as designated before executing the test setup.
- 3. A Test Setup Identifies What Data is Returned and How It Is Labeled** - A source unit conventionally sources voltage and measures current, or sources current and measures voltage. Win4145 provides the capability to return both voltage and current regardless of which quantity is forced. When a test setup combines *swept* sources with *stepped* sources, Win4145 can record an entire family of data vectors. For example, a source unit may apply a voltage sweep across the drain of a MOSFET and return both the voltage sweep and the measured current, while a different source unit applies a voltage bias to the gate. After the first source unit sweeps the drain, the source unit at the gate will step to a new bias voltage in preparation for a second drain sweep. The result of this test setup is a family of drain current data vectors labeled I_{ds1} , I_{ds2} , I_{ds3} , etc. Each data vector corresponds to a single curve.

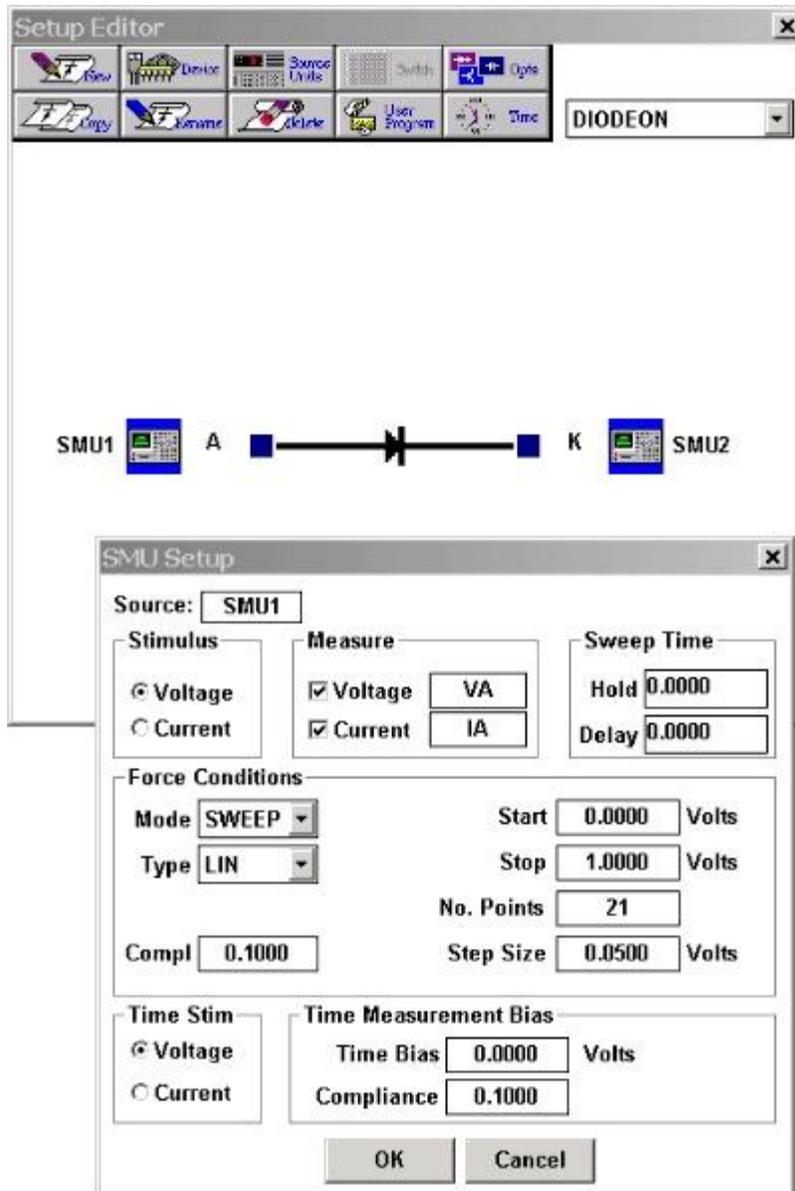


Figure 2-1: The test setup stores instrument setup conditions, documents the instrument/DUT connections, and stores what data is returned and how it is labeled.

THE TEST SETUP/PROJECT FILE RELATIONSHIP

A test setup defines the instrument configurations for a specific device measurement. Test setups are created in the Setup Editor. A project file stores one or more test setups as a single unit. Project files are defined in the File Manager. A project file will normally contain all of the test setups corresponding to a single device or wafer.



All of the data associated with a test setup is written to a single data window spreadsheet. Each data window spreadsheet corresponds to a single test setup. It is not recommended that the user create more than ten test setups in a single project file. If this limit is exceeded the stability of the operating system may deteriorate.

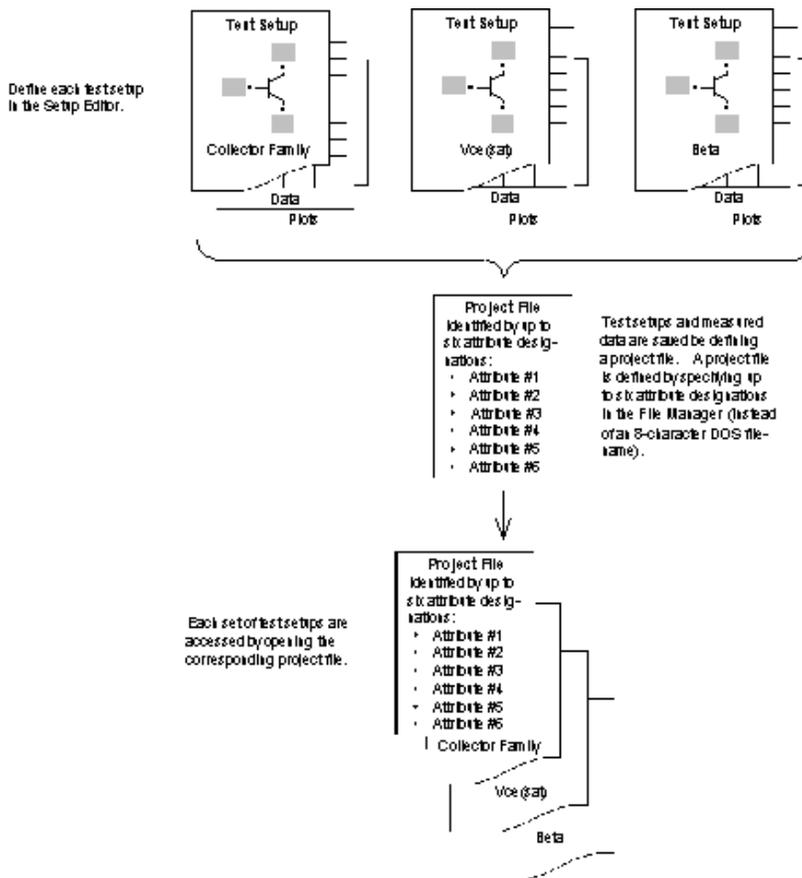


Figure 2-2: One or More Test Setups make up a Single Project File. A Test Setup is only Accessible from the Project File in which it was Created.



Test setups are only accessible from the project file in which they were created. A test setup cannot be copied and moved to another project file. The exception to this rule is when using a Save As command.

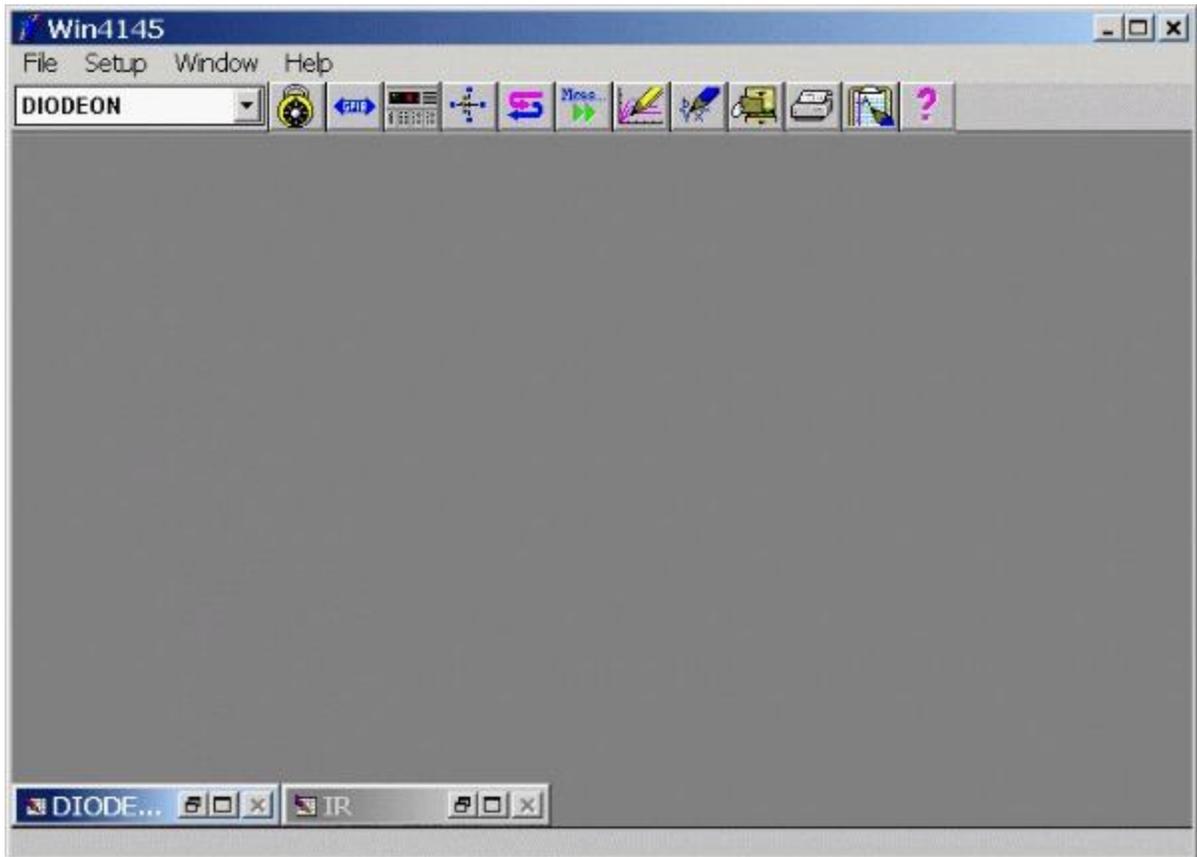


Figure 2-3: Test Setups corresponding to a Project File are displayed along the Bottom of the Desktop.

TEST SETUP APPLICATIONS

There are two general applications for which a test setup can be created. Each application is described by an example.

Consider the need to characterize the performance of a device under different electrical and environmental conditions. You may create a group of test setups in

which each test setup represents a different set of characteristic curves. For example, one test setup may measure drain family characteristics, another gate transfer, and still another sub-threshold drain characteristics.

Alternatively, consider the need to screen a series of devices within a module on a wafer. Use of a manual wafer probe would allow the movement of the probes between measurements. By using the device name as an attribute in the Project File name you may create a test setup which measures a device characteristic, such as VCE(sat). In this case each project file corresponds to a unique device.

THE SETUP EDITOR

Each test setup is uniquely created, named, edited, or deleted with the use of the Setup Editor. The Setup Editor is a graphic environment that is completely icon- and menu-driven.

SETUP EDITOR FUNCTIONS

1. Specify the instrument setup configurations of each source unit.
2. Document the connections between the instrument and device.
3. Specify what data is returned and how it is labeled.

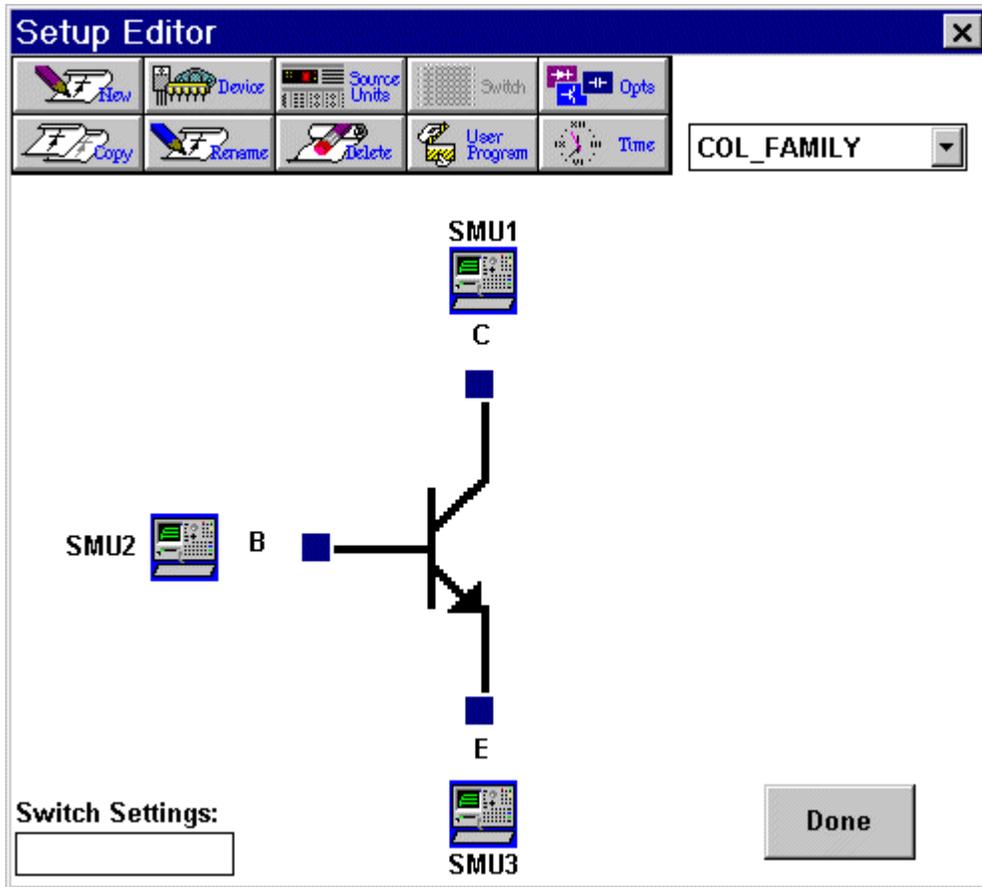


Figure 2-4: The Setup Editor is Used to Document Instrument/DUT Connections and Configure Instrument Setup Conditions.

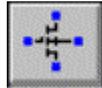
A test setup specifies the instrument configuration necessary to execute a device measurement. Each test setup is named by the user and specifies the test instrument, device type, source unit connections, and the setup conditions of each instrument source unit used in the test setup. A test setup is a collection of instructions specific to a single device measurement; therefore, a project file may contain numerous test setups.

OPENING THE SETUP EDITOR

A test setup is created or edited with the use of the Setup Editor. The Setup Editor is an icon-driven workspace that allows you to construct a test setup solely with the use of point and click editing.

HOW TO OPEN THE SETUP EDITOR:

1. Click the Setup Editor tool bar button, or
2. Select MEASUREMENT/EDIT SETUP from the measurement mode menu bar.



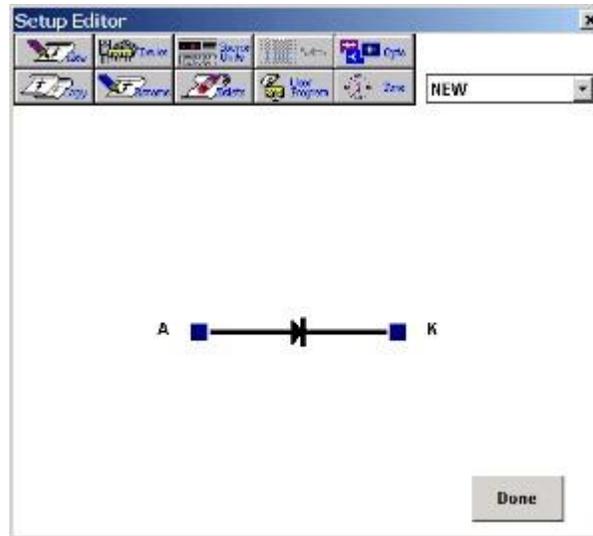
The Setup Editor can be opened by clicking the Setup Editor tool bar button.

The most obvious feature of the Setup Editor is the device schematic. The device schematic is a graphic representation of the component family selected for testing. The device schematic also includes a text field at each terminal location. The text fields display a lead or pin abbreviation. The lead or pin abbreviations cannot be edited.



The top right-hand corner of the Setup Editor includes a scroll box that displays the name of the current test setup. Clicking the scroll arrow will display a list of all the test setups defined in the current project file. This allows the user to rapidly switch between setups by choosing the desired setup name from the list.

When creating a test setup in a new project file, the drop-down list will be empty. The remaining buttons along the top of the Setup Editor dialogue box are used for creating and editing test setups.



This view shows the Setup Editor dialog box after the specification of a setup name. No source unit/DUT connections have been designated.

SELECTING A DEVICE SCHEMATIC

The most significant feature of the Setup Editor is the device schematic. The device schematic is intended to represent the layout of the instrument's test fixture.



By selecting the appropriate device schematic, the user can designate the hardware connections between the instrument and the device. This allows the user to create a visual record of the hardware configuration required for the corresponding test setup.

While it is recommended that the user select a device schematic corresponding to the layout of the instrument's test fixture, it is not necessary. The visual presentation of the Setup Editor is designed to document the hardware arrangements required for the corresponding device measurement.



There is no active relationship between the Setup Editor display and the actual connections between the instrument test fixture and the device. Win4145 actuates the source units according to the setup configurations specified by the user, regardless of how the source units are connected to the DUT or test fixture.

For example, you may select a BJT schematic and designate two source unit connections: one at the collector and another at the emitter. In reality, the source units designated in the Setup Editor may be cabled to each end of a diode test fixture. As long as the source/measure configuration of each source unit is configured for a diode measurement, it makes no difference what is visually designated in the Setup Editor display. The resultant data will correspond to the diode test.

A source unit connection must be designated in the Setup Editor in order to access the dialogue box used to control the source/measure configuration of the respective source unit. Therefore, the device schematic selected in the Setup Editor must include enough terminals to accommodate the number of source units required for the device measurement.



As a default the Setup Editor will load the MOSFET device schematic each time a new test setup is created. If necessary, select a different device schematic by clicking the Setup Editor DEVICE button.



Please note that if you change the device in a setup after you have added and configured the Source units you will lose all setup information.

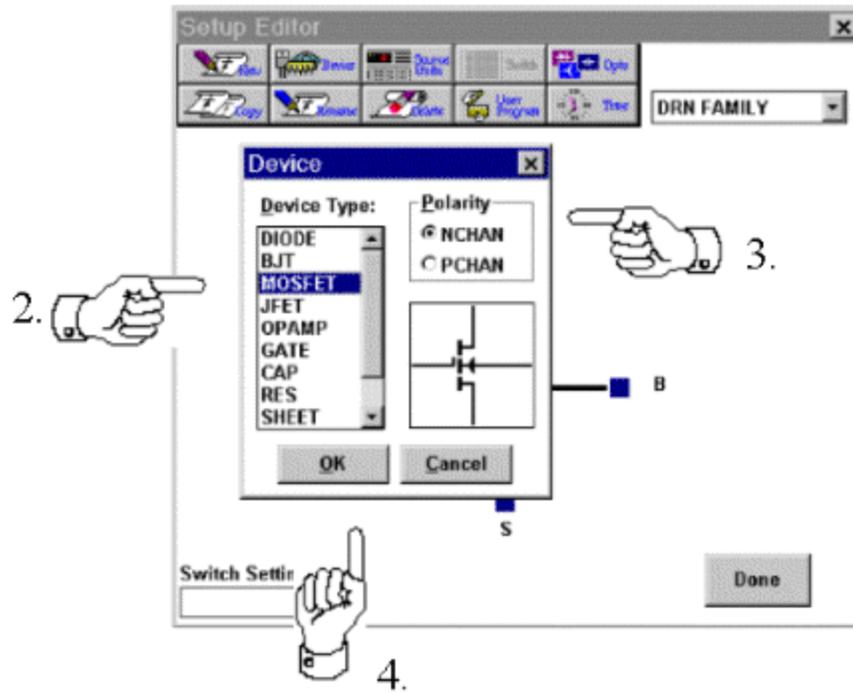
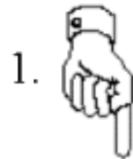


Figure 2-5: How to Select the Device Schematic.

HOW TO SELECT A DEVICE SCHEMATIC:

1. Click the DEVICE button on the right-hand side of the Setup Editor. Clicking the DEVICE button will open the Device dialogue box.
2. Highlight the desired DEVICE TYPE.
3. Click on the appropriate POLARITY choice, if applicable.
4. Click the OK button. This will close the Device dialogue box and restore control to the Setup Editor. The Setup Editor will display the selected device schematic.



There are more than twelve different device schematics available in the Setup Editor Device dialogue box. The device schematics are visual aids only; there is no connection between the selected device schematic and the physical arrangement of the instrument's test fixture.

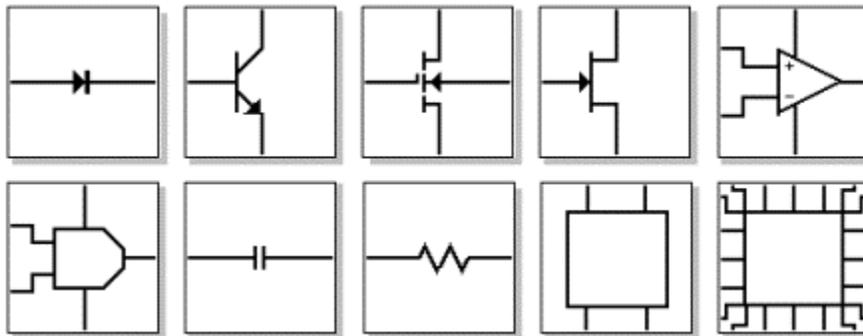


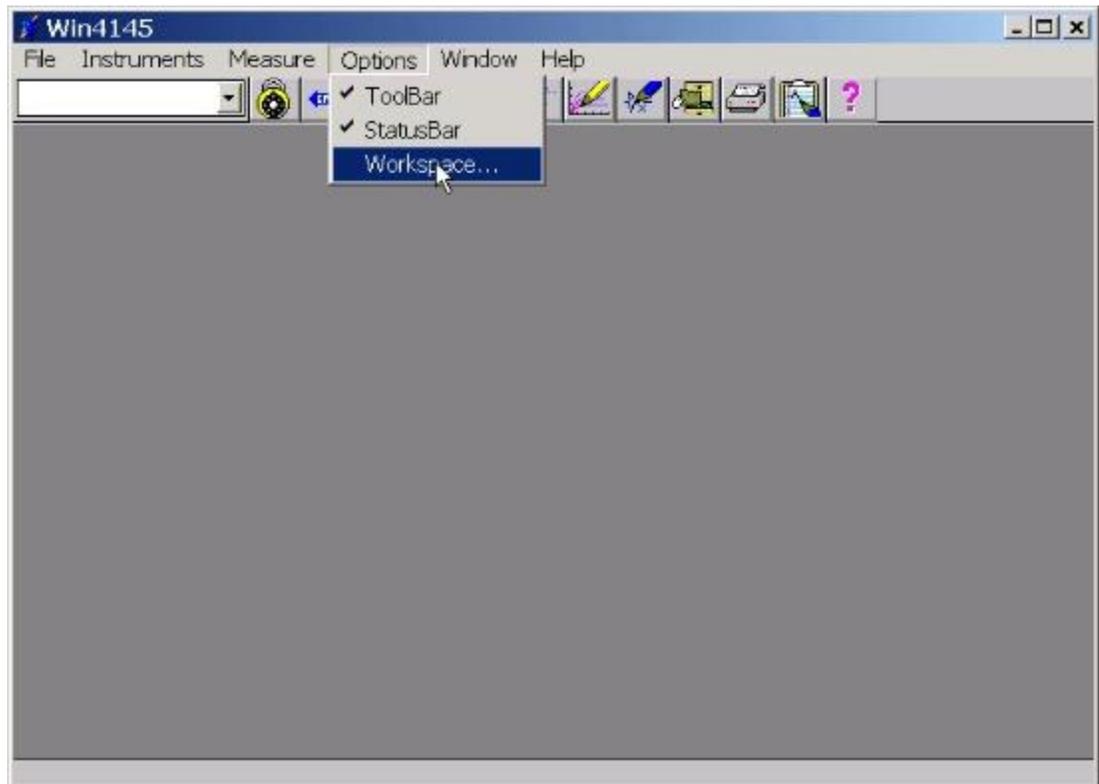
Figure 2-6: A sample of the Device Schematics that can be selected from the Device dialog box.

SPECIFYING THE DEFAULT DEVICE SCHEMATIC

As a default, the Setup Editor will load the MOSFET schematic each time a test setup is created. If the default schematic is not appropriate, select an alternative schematic by clicking the Setup Editor DEVICE button. If desired, the user can designate any of the available device schematics as the default selection.



The default device schematic is designated in the Workspace dialogue box.



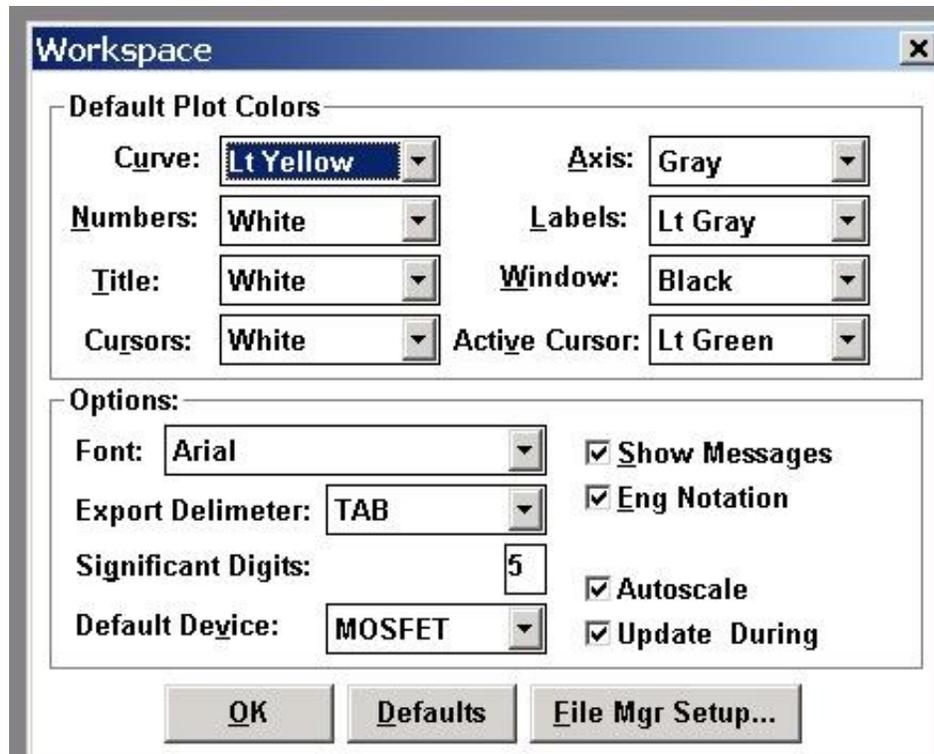


Figure 2-7: How to Specify the Default Device Schematic

HOW TO SPECIFY THE DEFAULT DEVICE SCHEMATIC

1. Select OPTIONS/WORKSPACE from the measurement mode menu bar. This will open the Workspace dialogue box.
2. Select the desired schematic designation in the Default Device field. Click the scroll arrow in the Default Device field to display a list of available options. Click on the desired designation.
3. Click OK. The new designation will take effect the next time the Setup Editor is displayed.

DESIGNATING INSTRUMENT/DUT CONNECTIONS

The connections between the instrument and the device can be designated in the Setup Editor. While it is recommended that the connections designated in the Setup Editor correspond to the reality of your hardware arrangement, it is not necessary.



The visual presentation of the Setup Editor is designed to document the hardware arrangements required for the corresponding device measurement. However, there is no active relationship between the Setup Editor display and the actual connections between the instrument test fixture and the device. Win4145 actuates the source units according to the setup configurations specified by the user, regardless of how the source units are connected to the DUT or test fixture.

For example, you may select a MOSFET schematic and designate two source unit connections: one at the drain and another at the gate. In reality, the source units designated in the Setup Editor may be cabled to each end of a diode test fixture. As long as the source/measure configuration of each source unit is configured for a diode measurement, it makes no difference what is visually designated in the Setup Editor display. The resultant data will correspond to the diode test.

Connections are designated by first selecting a source unit from the Source Units dialogue box. After the source unit is selected, click the blue pad next to the desired schematic pin. An instrument icon, along with the name of the connected source unit, will appear above the schematic pin indicating the connection.

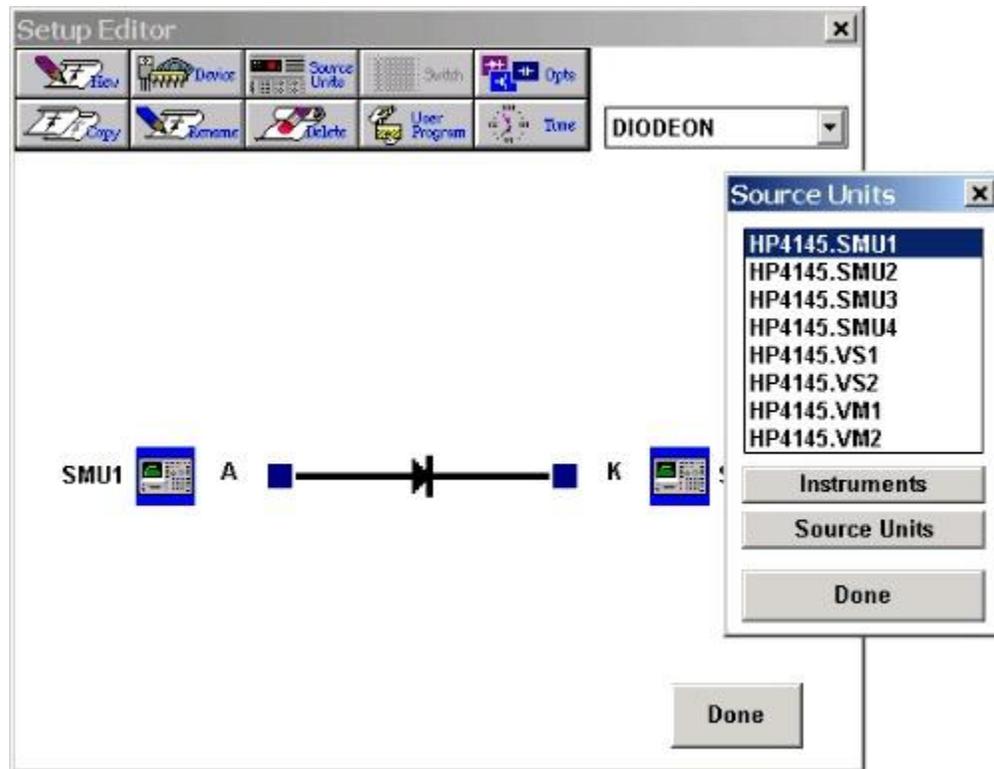


Figure 2-8: How to Designate a Connection Between the Instrument and the DUT.

HOW TO DESIGNATE THE INSTRUMENT/DUT CONNECTIONS:

1. Open the Source Units dialogue box from the Setup Editor. The Source Units dialogue box will display a list of available instrument supplies
2. Highlight the desired source unit.
3. Designate a connection by clicking on the blue pad next to the desired schematic pin. An instrument icon will appear above the connection.
4. Two source units can be assigned to a single pin. To designate a second source unit connection, repeat Steps #3 and #4. Both source units must be from the same instrument. When a pair of source units are assigned to the same schematic pin, the name of the source unit connected first will be listed above the instrument icon.

REMOVING INSTRUMENT/DUT CONNECTIONS

If the connection between an instrument source unit and the device is designated in error, the Setup Editor designation can be disconnected. To disconnect a designated instrument connection, select the appropriate source unit identity in the Source Units dialogue box. After the source unit is identified, click the blue pad next to the schematic pin where the source unit connection is mistakenly attached. Win4145 will display a message box asking you to confirm your desire to remove the connection. Click YES to complete the procedure.

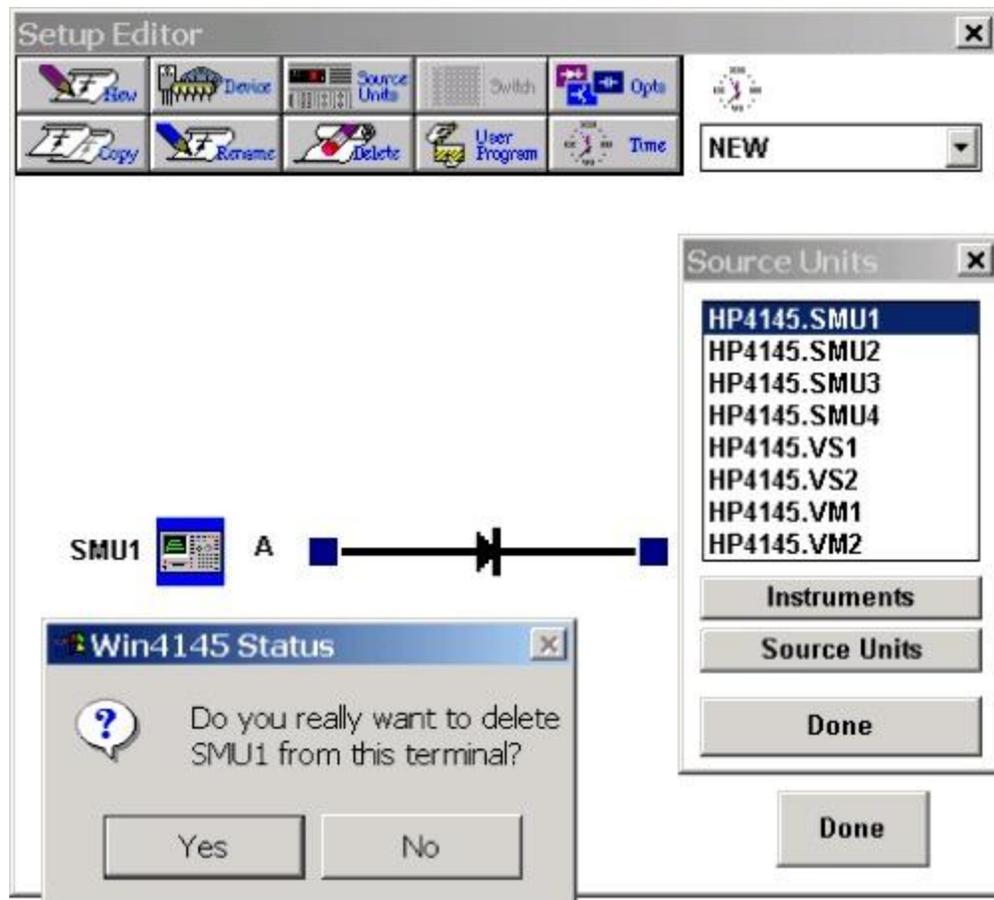


Figure 2-9: How to Remove an Instrument/DUT Connection

HOW TO REMOVE AN INSTRUMENT/DUT CONNECTION

1. Click the Setup Editor SOURCES button. This will open the Source Units dialogue box. The available source units and instrument supplies will be displayed.
2. Select the identity of the source unit or supply you wish to disconnect from the Setup Editor schematic.
3. Click the blue pad next to the schematic pin where the source unit connection is mistakenly attached. If more than one source unit is connected to the selected schematic pin, click the instrument icon to display a list of attached source units.
4. Win4145 will display a message box asking you to confirm your desire to remove a connection. Click YES to complete the procedure and restore control to the Setup Editor.

DEFINING SOURCE UNIT SETUP CONFIGURATIONS



The setup configuration of each source unit is controlled from the Source Unit Setup dialogue box. Each source unit has its own Setup dialogue box. The content and layout of each Source Unit Setup dialogue box varies according to the different instrument models.

For detailed instructions that explain how to configure a source unit, please refer to the chapter of this manual corresponding to the specific instrument driver in question.

HOW TO DISPLAY A SOURCE UNIT SETUP DIALOGUE BOX:

1. In order to display a Source Unit Setup dialogue box, the corresponding source unit must be attached to one of the device schematic pins displayed in the Setup Editor. If the source unit is not connected, refer to *Designating Instrument/DUT Connections*.
2. If only one source unit is connected to the device schematic pin, click once on the instrument icon. If more than one source unit is connected, proceed to Step #2.
3. If more than one source unit is connected to the device schematic pin, click once on the instrument icon to display a list of the source units connected to the selected pin.
4. Highlight the desired source unit by clicking the source unit name once. Double-click on the highlighted source unit to open the corresponding Source Unit Setup dialogue box.

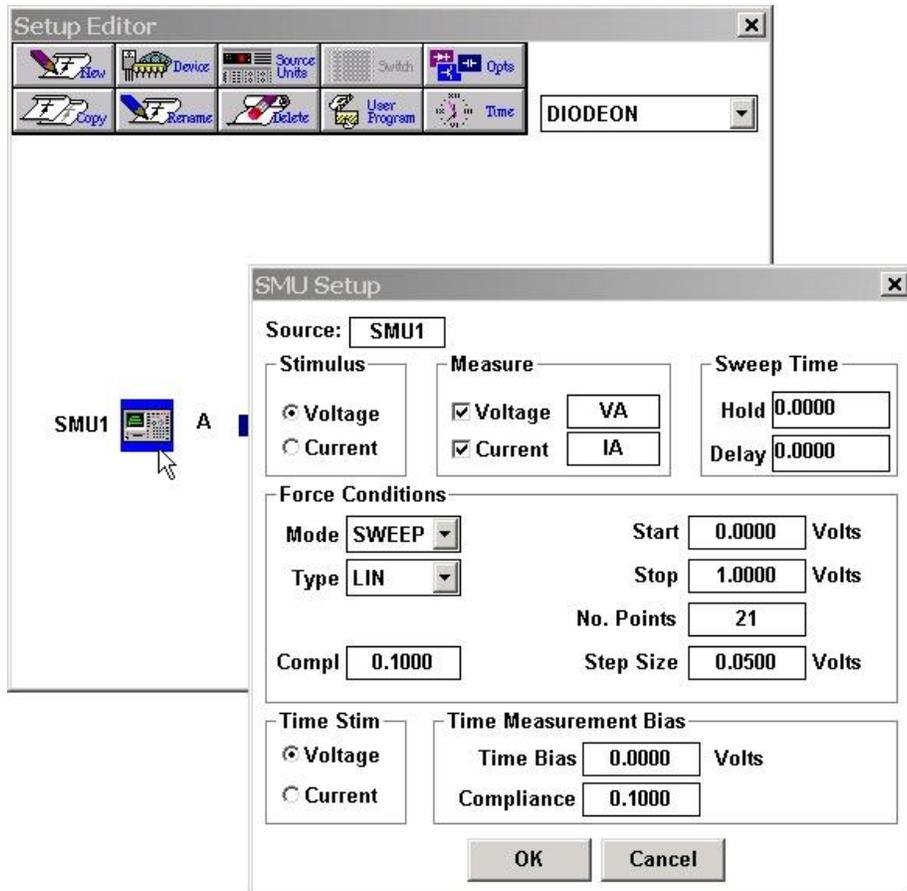


Figure 2-10: How to Display a Setup Dialogue Box When a Single Source Unit is Connected to the Device Schematic Pin.

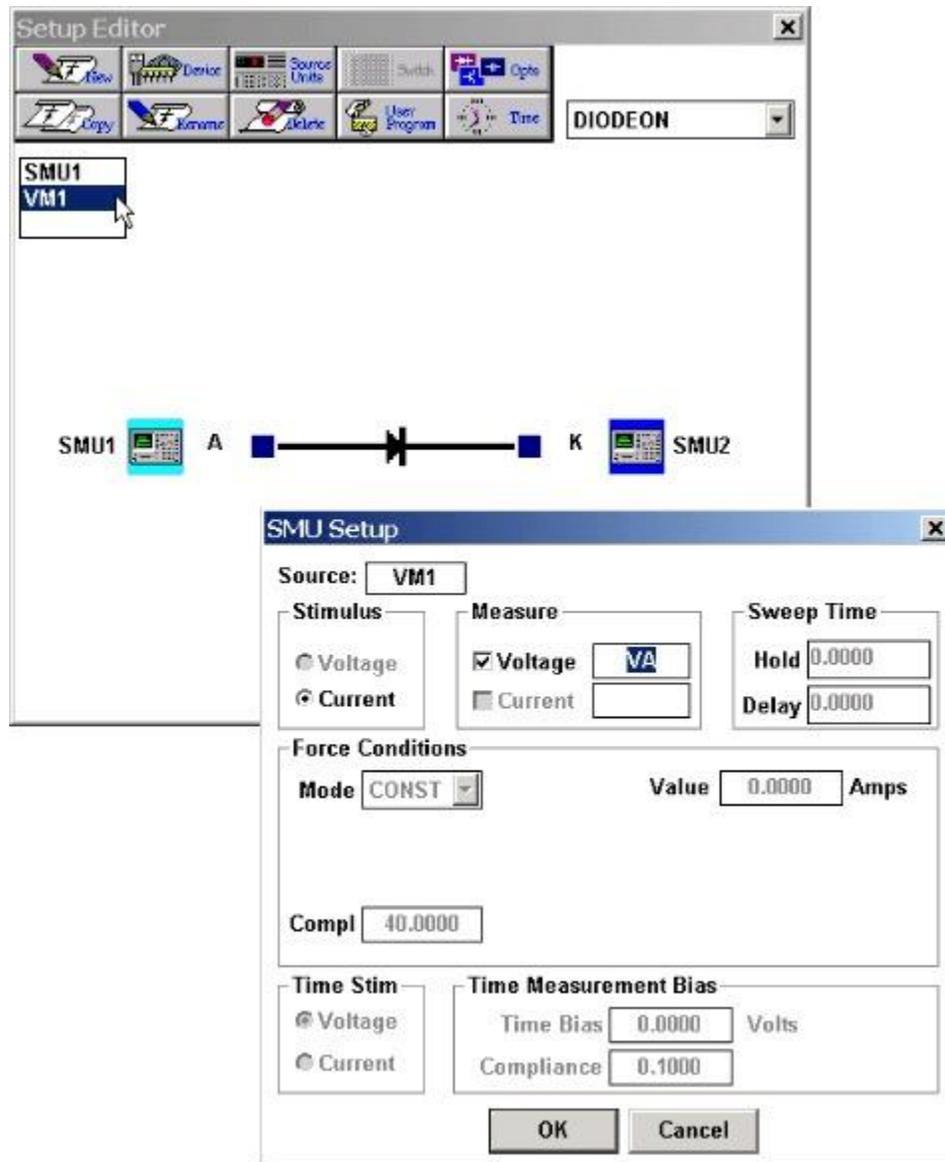


Figure 2-11: How to Display a Setup Dialogue Box When Multiple Source Units are Connected to the Device Schematic Pin.

CREATING A SINGLE TEST SETUP

Below is a description of the test setup definition process. Each step listed below is addressed in greater detail in the sections that follow.

HOW TO CREATE A TEST SETUP:

1. Open the Setup Editor. Click the NEW button and specify a test setup name at the prompt.
2. Click the DEVICE button and choose a device schematic that corresponds to the Device Under Test (DUT).
3. From the Setup Editor, open the Source Units dialogue box by selecting the SOURCES button. Designate the connections between the instrument source units and the device.
4. Configure the source/measure controls of each source unit connected to the device.

CREATING A TIME DOMAIN TEST SETUP

CONTRASTING THE HP4145 AND ICS TIME DOMAIN UTILITIES

The functionality of the HP4145 Time Domain utility is similar to ICS' Time Domain Time Measure mode with some exceptions. The HP4145 Time Domain utility is limited to 500 or 1024 data points (HP4145A or HP4145B respectively) measured at a maximum interval of 10 seconds. In other words, a time sweep configured with the HP4145 Time Domain utility is limited to a maximum duration of either 1 hour and 23 minutes or 2 hours and 50 minutes depending upon the version of the HP4145. However, the Time Measure mode configured with the ICS Time Domain utility has no practical limit. The maximum time sweep duration that can be specified in the ICS Time Domain utility is on the order of 1×10^{13} hours.

The ICS Time Domain utility provides a second measurement mode: Bias Measure. In Bias Measure mode, a device can be biased at constant conditions for a designated length of time prior to the execution of the test setup. The HP4145 Time Domain utility does not provide this capability.

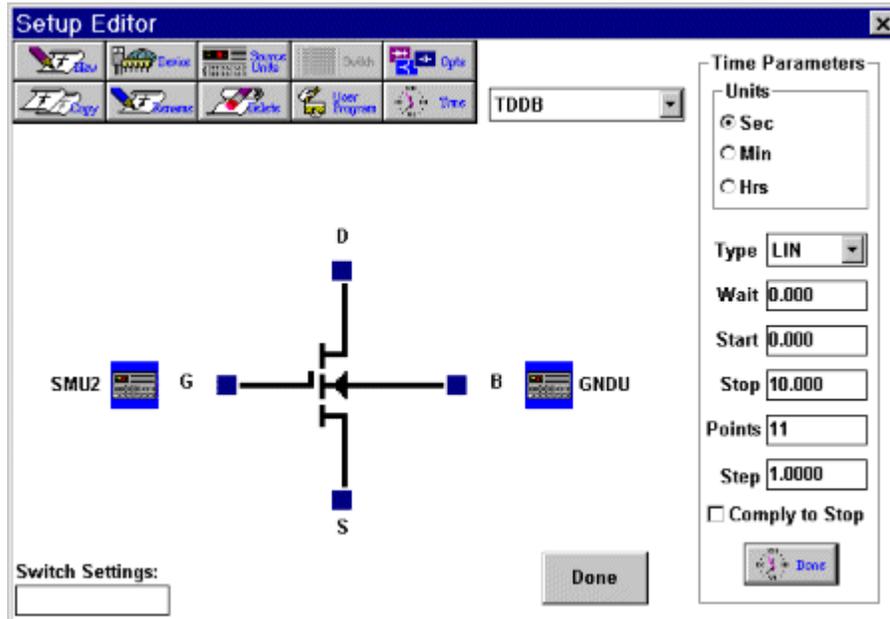
The HP4145 Time Domain utility allows the user to generate a family of time sweep curves by configuring one of the source units in STEP mode. The ICS Time Domain utility does not support multiple time sweeps in the same test setup.

The table below compares the capability of the HP4145 and ICS Time Domain utilities.

Capability	HP4145 Firmware	ICS
Maximum Time Sweep Duration	1 hr 23 min (4145A) 2 hr 50 min (4145B)	Unlimited (1×10^{13} hr)
Supported Modes	Time Measure	Time Measure Bias Measure
Multiple Time Sweeps	Yes	No
Control Location	HP4145 Mainframe Setup Dialogue Box (Setup Editor OPTIONS Button)	Time Setup Dialogue Box (Setup Editor TIME Button)

Table 1: A Comparison of the HP4145 and ICS Time Domain Utilities

WIN4145 TIME DOMAIN CONTROLS



Time domain capability allows the user to include time in the definition of a test setup. For example, the user can specify a bias delay in which a constant bias signal is applied to the device under test. After the specified length of time, the bias signal is removed and the device measurement is executed. Alternatively, the user can apply a constant signal to the device under test and return a device measurement at designated intervals of time. Time domain capability is an enhancement to the functionality of most instruments. Many of the instruments supported by Win4145 do not provide this capability when controlled from the front panel.

The time domain utility is normally turned OFF. The time domain utility is turned ON by selecting a measurement mode of Bias Delay or Time Measure in the Measurement Remote Control dialogue box. The Setup Time controls are made visible by clicking the Time button in the Setup Editor. When the time domain utility is implemented in a test setup, the time domain icon will appear in the upper right of the Setup Editor.



The time domain utility can be configured in one of three possible modes: Bias Delay mode, Time Measure mode, or Sequence Stress mode.

Bias Delay mode allows the user to stress the device with a constant bias signal before performing a single device measurement. From the perspective of a source unit, the Bias Delay mode consists of two parts: the bias delay and the device measurement. Thus there is a point between the bias delay and the measurement where the device is NOT under bias.



In Bias Delay mode, each source unit setup includes two different setup configurations: Time Measurement parameters used to designate the constant bias signal applied during the bias delay, and the standard measurement configuration used to designate the source unit setup for the post-delay device measurement.

Time Measure mode allows the user to stress the device with a constant bias signal while measuring the device at designated time intervals. In Time Measure mode, the same source unit configuration is applied to the device for the duration of the test setup.

The setup configuration of each source unit, for Bias Delay and Time Measurements, is defined in the Time Measurement Bias Group of controls in the respective Source Unit Setup dialogue box.

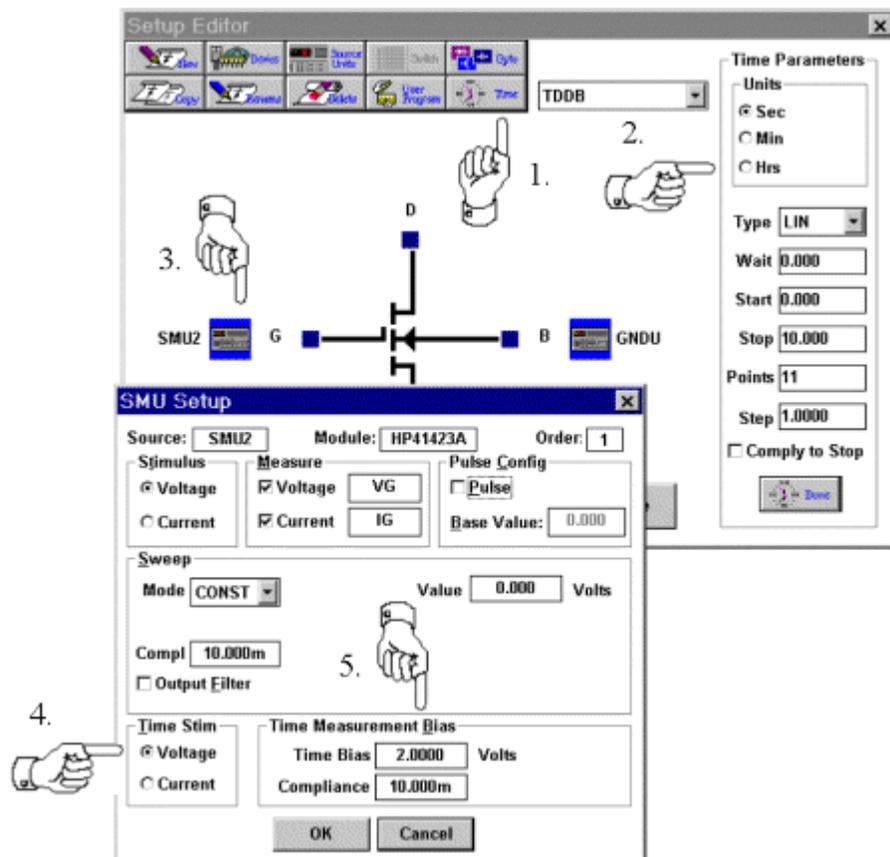


Figure 2-12: Configuring Source Units for Win4145 Time Measurements.

HOW TO SETUP WIN4145 TIME MEASUREMENT

PARAMETERS:

1. Display the Win4145 Time Measurement controls by clicking on the TIME button.
2. Specify the Bias Delay or Time Measurement time parameters in the TIME PARAMETERS controls.
3. Open the Source Unit Setup dialogue box by clicking on the source unit icon.
4. Specify the Time Measurement source stimulus in the TIME STIM controls for Bias Delay or Time Measurement modes.
5. Specify the Time Measurement bias and compliance conditions in the TIME MEASUREMENT BIAS controls for Bias Delay or Time Measurement modes.
6. Specify standard measurement parameters for Bias Delay mode.



Source unit outputs are limited to constant signals during the Time Measure mode and during the bias delay portion of the Bias Delay mode. There are no restrictions to the source unit configurations applied during the device measurement portion of the Bias Delay mode.



The minimum time that can be specified in any of the timing fields is a function of the instrument and specific measurement configuration. If any time specification is less than 100ms, Win4145 will return an warning message stating that the time interval may not be met.

The designations in any of the timing fields can be specified with engineering units, such as "1m" representing "1 milli-". The multiplier represented by the engineering unit will be applied to the scale designated at the top of the Setup Time dialogue box. For example, if the time scale is configured in seconds, a "1m" specification in any timing field represents 1 millisecond. If the time scale is configured in hours, the "1m" specification represents "1 millihour". To avoid any confusion, consider limiting the use of engineering units to test setups in which the time scale is configured in seconds.

Time domain test setups can be saved in project files that include static test setups. When designating a test setup sequence, both time domain and static test setups can be designated in the same test setup sequence.

BIAS DELAY MODE

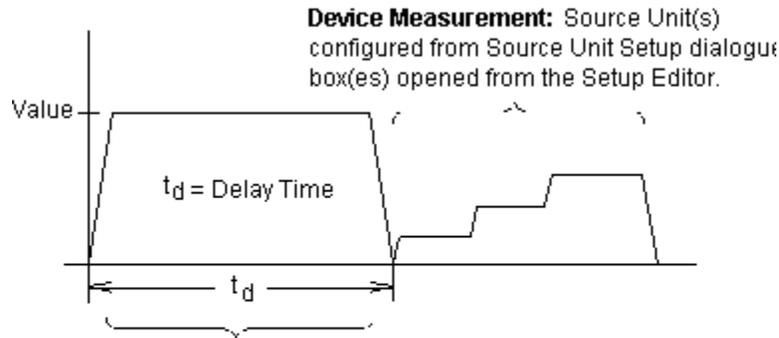
The Bias Delay mode allows the user to stress the device at a constant bias before the device measurement is performed. When the time utility is configured in Bias Delay mode, the user must specify a delay time.



During the bias delay time, each designated source unit will apply a constant signal to the device under test. After the delay time, the device is de-biased and the device measurement is performed as defined in the Setup Editor. A typical source unit output signal for a source unit controlled in Bias Delay mode is shown on the following page.

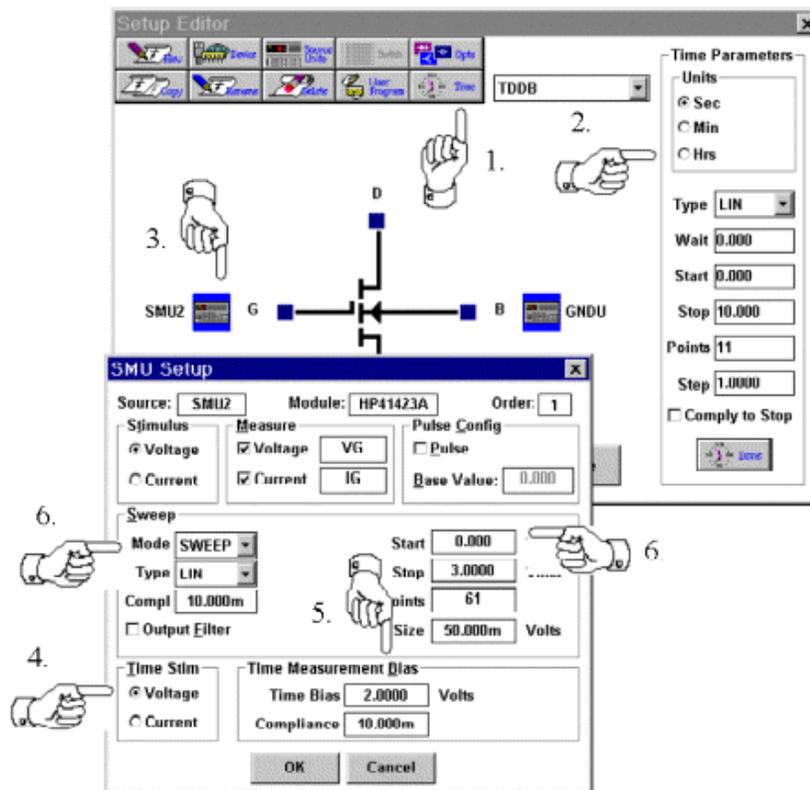
For example, if the same SMU is used to force a constant bias during the delay and generate a ramped sweep during the device measurement, there will be a discontinuity in the SMU output when the ramped sweep is configured to start at the previous bias value.

When the time domain utility is configured in Bias Delay mode, each Source Unit Setup dialogue box can store two different setup configurations: one configuration for the bias delay portion of the test setup, and another for the device measurement portion of the test setup.



Bias Delay: Source Unit(s) configured from Source Unit Setup dialogue box(es) opened from the Setup Time dialogue box.

Figure 2-13: An Example of a Source Unit Output Signal Configured in Bias Delay Mode.



HOW TO SETUP WIN4145 BIAS DELAY MEASUREMENT PARAMETERS:

1. Display the Win4145 Time Parameters control by clicking on the TIME button.
2. Specify the Bias Delay time parameters in the TIME PARAMETERS control.
3. Open the Source Unit Setup dialogue box by clicking on the source unit icon.
4. Specify the source stimulus in the TIME STIM controls for the constant bias portion of the Bias Delay mode.
5. Specify the Time Measurement bias and compliance conditions in the TIME MEASUREMENT BIAS controls for the constant bias portion of the Bias Delay mode.
6. Specify standard measurement parameters for the measurement portion of Bias Delay mode by completing the Sweep, Stimulus, and Measure controls of the SMU Setup.

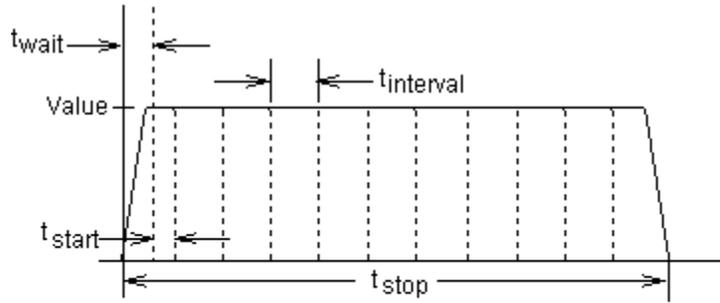
TIME MEASURE MODE

Time Measure mode allows the user to stress the device with a constant bias signal while measuring the device at designated time intervals.



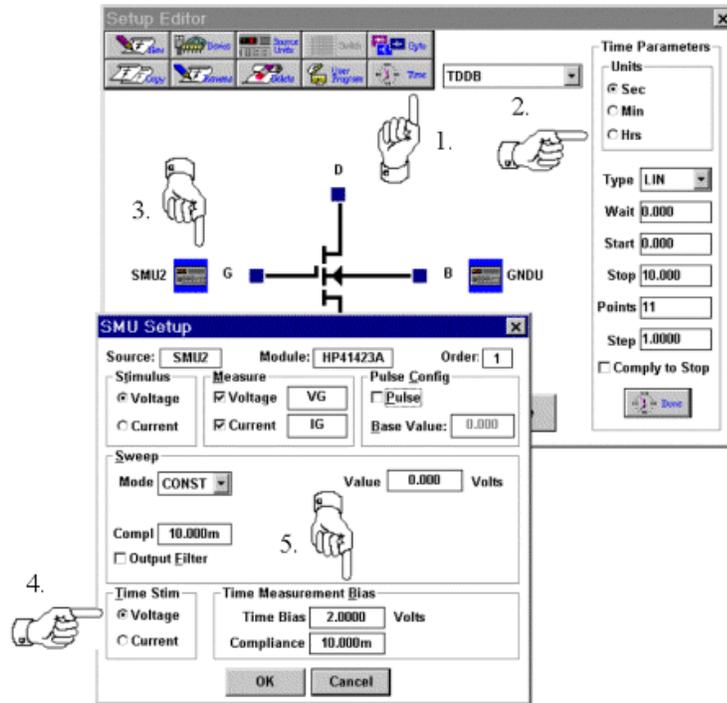
Instrument outputs WILL NOT be de-biased between measurement intervals.

In Time Measure mode, the same source unit configuration is applied to the device for the duration of the test setup. The setup configuration of each source unit must be configured in the corresponding Source Unit Setup dialogue box opened from the Setup Time dialogue box. Any setup conditions configured in the Source Unit Setup dialogue boxes opened from the Setup Editor will be ignored. A typical source unit output signal for a source unit controlled in Time Measure mode is shown below.



Time Measure:Source Unit(s) configured from Source Unit Setup dialogue box(es) opened from the Setup Time dialogue box. Any setups configured from the Setup Editor will be ignored.

Figure 2-14: An Example of a Source Unit Output Signal Configured in Time Measure Mode.



HOW TO SETUP WIN4145 TIME MEASUREMENT PARAMETERS:

1. Display the Win4145 Time Measurement controls by clicking on the TIME button.
2. Specify the Time Measurement parameters in the TIME PARAMETERS controls.
3. Open the Source Unit Setup dialogue box by clicking on the source unit icon.
4. Specify the Time Measurement source stimulus in the TIME STIM controls.
5. Specify the Time Measurement bias and compliance conditions in the TIME MEASUREMENT BIAS controls.

TIME MEASURE CONTROLS



Please note that the actual measurement during the Time Measurement execution takes a finite amount of time, thus the total time for test execution may exceed the total time specified in the Time Parameters control. This may be avoided by checking the Comply to Stop button at the bottom of the Time Parameters control. By doing so the test will end at the specified stop time regardless of whether all data points have been taken.

When the time domain utility is configured in Time Measure mode, a group of specification fields will be displayed in the Setup Time dialogue box that are relevant to the Time Measure mode only.

TYPE

The Type field is used to designate whether the measurement interval is linearly or logarithmically calculated. Click the Type field scroll-arrow to display a list of the two options. Click on the desired designation. If a logarithmic measurement interval is designated in the Type field, the Interval field is unavailable since the specification of an interval time is no longer relevant. The log type uses the start and stop times along with the number of points to determine the number of time periods per decade.

WAIT TIME

Wait time is specified in the Wait field. The wait time is the length of time the instrument will wait at bias before measuring the start time. Specifying a wait time allows the user to soak the device at the constant bias level before initiating any measurements.

START TIME

The Start field is used to designate the length of time that will elapse from the end of the wait time to the initiation of the first measurement. Instrument outputs are held at the bias value throughout the duration of the START time. If both WAIT and

START values are specified, the total soak time at bias before the first measurement is obtained is equal to the sum of the WAIT and START times. No debiasing occurs in the transition between WAIT time and START time. If a logarithmic measurement interval is specified in the Type field, a "100m" default specification will appear in the Start field.

STOP TIME

The Stop field is used to designate the duration of the test setup. If the Interval field is edited after all of the Time Measure fields are specified, the Stop field will be updated automatically to accommodate the change.

POINTS

The Points field is used to specify the total number of measurements that will be performed throughout the duration of the test setup. If a linear measurement interval is specified in the Type field, configuring the Points field will automatically calculate the measurement interval displayed in the Interval field.

MEASUREMENT INTERVAL

The Interval field is used to specify the length of time between consecutive measurements. Instrument outputs WILL NOT be debiased between measurement intervals. The Interval field is only displayed if a linear measurement interval is designated in the Type field. The Interval field is automatically calculated once the Start, Stop, and Points fields are specified. If the Interval field is edited, the Stop field will be updated automatically to accommodate the change.

COMPLY TO STOP

The Comply to Stop switch is used to specify whether Win4145 should stop the measurement at the stop time even if the number of points or measurements have not been completed.

CREATING MULTIPLE TEST SETUPS



It is not recommend that the user create more than ten test setups in a single project file. If this limit is exceeded the stability of the operating system may deteriorate.

HOW TO CREATE A SECOND TEST SETUP:

1. Click the NEW button located on the top left-hand side of the Setup Editor dialogue box. Win4145 will clear the active setup and open the New Setup dialogue box.
2. Enter a name in the New Setup dialogue box and click the OK button. This will close the dialogue box and load the default device schematic. The test setup name just specified will appear in the Setup Editor setup window.
3. Continue with Steps #2 through #4 listed under the instructions entitled *Creating a Single Test Setup* presented earlier in this section.

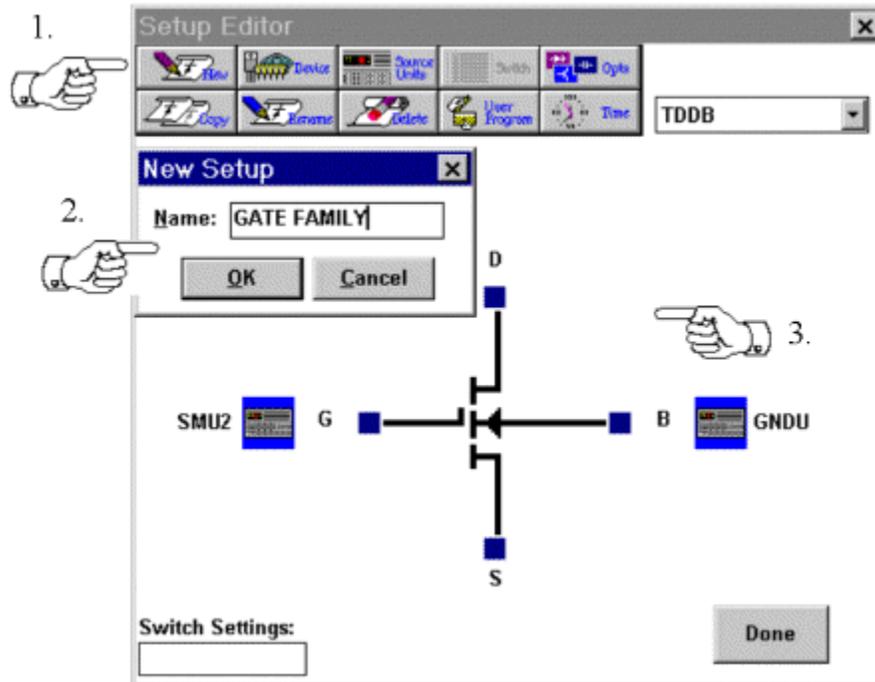


Figure 2-15: How to Create a New Test Setup.

DESIGNATING THE ACTIVE TEST SETUP

The active test setup is the test setup that will be executed when the instrument is actuated in Single Measure mode. (For a complete description of Single Measure and Sequence Measure modes, please refer to *Chapter 4: Measuring Devices*.) The active test setup can be designated by clicking once on the corresponding data window spreadsheet icon. Clicking once on a data window spreadsheet icon will also open a pop-up system menu. Click anywhere on the desktop to close the system menu.

The active test setup can also be designated from a list of project file test setups displayed in the toolbar or Setup Editor.



Figure 2-16: Each Data Window Spreadsheet Minimized At the bottom of the desktop corresponds to a Test Setup.

HOW TO DESIGNATE A TEST SETUP:

1. Both the toolbar and Setup Editor setup windows display the active test setup. Click the scroll arrow in either setup window to display a list of defined test setups.
2. Designate the active test setup by clicking once on the setup name. This will close the drop-down list and replace the contents of the setup window with the new selection.
3. If you choose not to change the active setup, click the scroll arrow again to make the list disappear.



Figure 2-17: How to Designate a Test Setup from the Tool Bar.

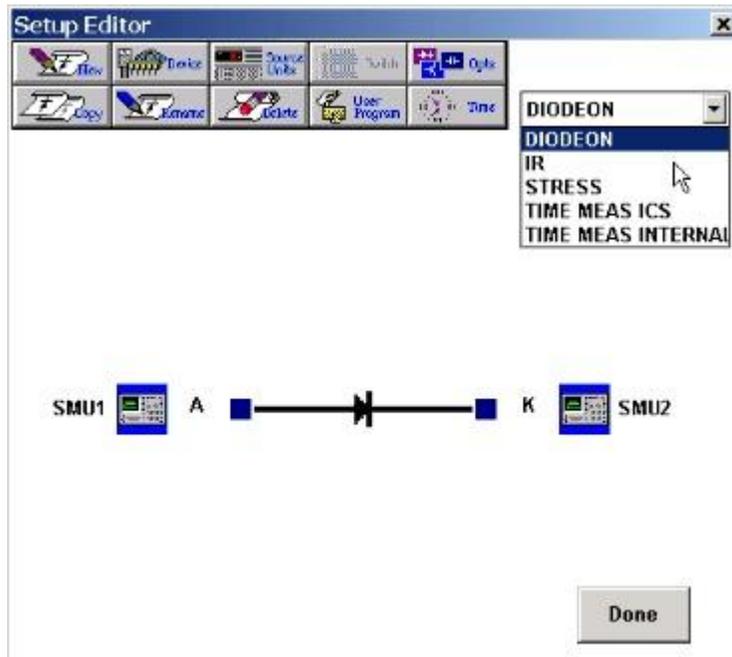


Figure 2-18: How to Designate a Test Setup from the Setup Editor.

SETUP EDITOR CONTROLS

DELETING A TEST SETUP



Test setups can only be deleted from the Setup Editor. Deleting a test setup will also delete the corresponding data window spreadsheet and any corresponding plot windows.



Figure 2-19: How to Delete a Test Setup.

HOW TO DELETE A TEST SETUP:

1. From the Setup Editor setup window, select the obsolete test setup.
2. Click the DELETE button along the right-hand side of the Setup Editor dialogue box.
3. Win4145 will require a confirmation before deleting the test setup. Answer YES or NO in the displayed message box.

COPYING A TEST SETUP



Test setups can only be copied within the same project file. Test setups CANNOT be copied to alternative project files.

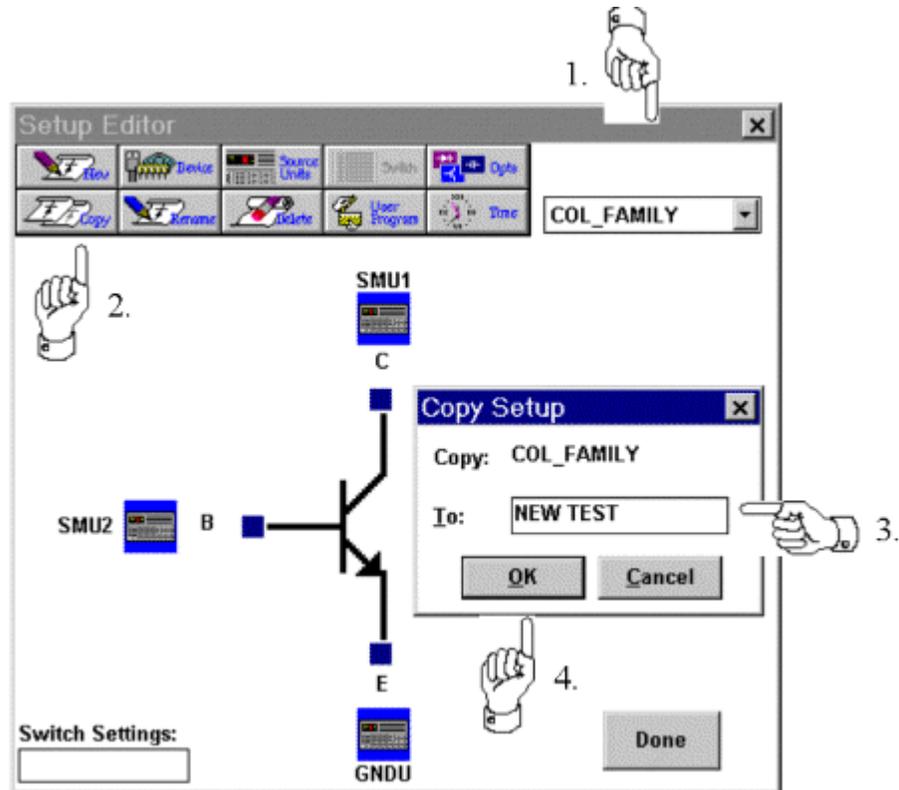


Figure 2-20: How to Copy a Test Setup.

HOW TO COPY A TEST SETUP:

1. From the Setup Editor setup window, select the desired test setup.
2. Click the Setup Editor COPY button. This will open the Copy Setup dialogue box.
3. Specify a unique setup name in the dialogue box text field.
4. Click the OK button to close the Copy Setup dialogue box and restore control to the Setup Editor. Only the designated instrument/DUT connections and the source unit setup configurations will be copied to the new test setup. The data and plot windows corresponding to the original test setup will not be copied.

RENAMING A TEST SETUP

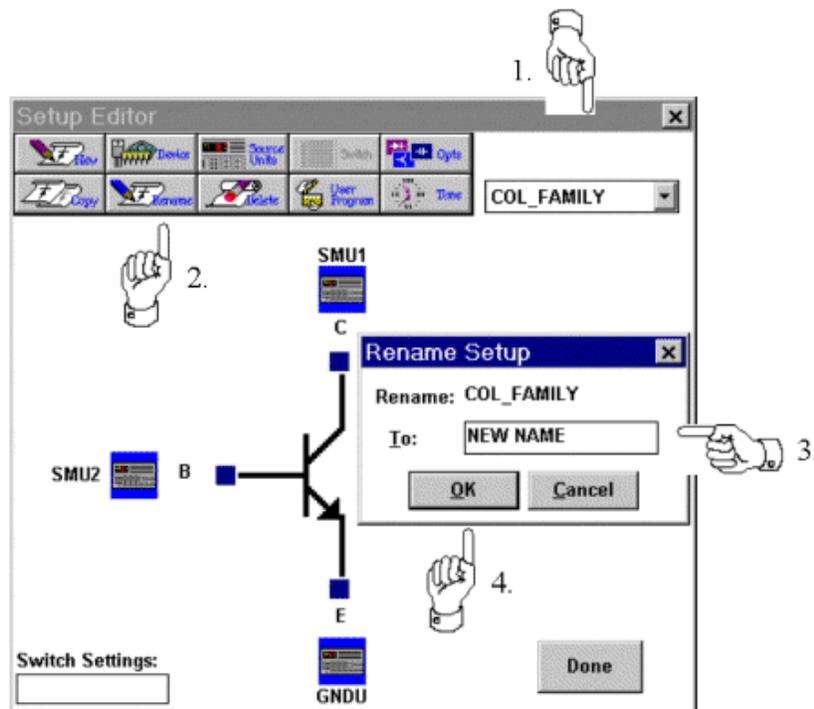


Figure 2-21: How to Rename a Test Setup.

HOW TO RENAME A TEST SETUP:

1. From the Setup Editor setup window, select the desired test setup.
2. Click the Setup Editor RENAME button. This will open the Rename Setup dialogue box.
3. Specify a unique setup name in the dialogue box text field.
4. Click the OK button to close the Rename Setup dialogue box and restore control to the Setup Editor. If an identically name plot window corresponds to the original title of the renamed test setup, the name of the plot window will not be updated.

EXECUTING EXTERNAL PROGRAMS



User or External Programs may be executed prior to and after a setup measurement is executed. User Programs may be written for unsupported instruments, hot chucks, and other operations.

User Program Execution

Execution Mode: Pre-measurement

Command Line: c:\probe\probe.exe -x2 -y9

Execution Timeout: Seconds Mode Value: 30

Options: Windows Program Type Show View

List of Commands to Execute: c:\hchuck\hchuck.exe -t200

List Edit Controls: Append Insert Edit Replace Cut Delete All

Done Save Command To List Clear Command

EXECUTION MODE

Execution mode sets a user/external program to execute before or after the measurement of your setup file. Select:

Pre-measurement - Perform user/external program before the measurement.

Post-measurement - Perform user/external program after the measurement

COMMAND LINE

Command line to be executed to run user/external executable program. You can specify up to 10 command lines to be executed before a measurement. You can specify up to 10 commands lines to be executed after a measurement.

EXECUTION TIMEOUT

Sets the timeout value for execute user/external program execution.

Mode- Selects unit for timeout time.

Value- Set desired value.

OPTIONS

Program Types- In order for Win4145 to execute an external program the program type must be known and designated in the Options fields.

DOS- a DOS file type

WINDOWS- a Windows program

PIF file- a Windows PIF file

View- Allows the user to select whether the program execution will be displayed.

Show- Show the external program executing.

Hide- Don't show the external program executing.

LIST OF COMMANDS TO EXECUTE

Lists the commands executed with the current setup. The commands are executed in the order listed in this table.

List Edit Controls- These controls allow editing, deleting, and re-ordering of items in the list.

Append- Appends the command cut into memory to the end of the list.

Insert- Inserts the command cut into memory at the location of the currently selected command.

Edit- Places the currently selected command into the command line field for editing. After editing the command must be saved again.

Replace- Replaces the currently selected command with the command cut into memory.

Cut- Cuts the currently selected command into memory.

Delete All- Deletes all commands from the list.

HOW TO DEFINE A USER PROGRAM FOR EXECUTION:

1. From the Setup Editor setup window, select the USER PROGRAM button to display the User Program Execution dialogue box.
2. Specify whether the user program will be executed before or after the measurement by selecting either PRE-MEASUREMENT or POST-MEASUREMENT from the EXECUTION MODE control.
3. Specify the name of the user program to be executed in the COMMAND LINE field. The name of the user program must include a full path and command line options necessary to execute the program.
4. Specify the execution timeout mode in the MODE field and the timeout value in the VALUE field.
5. Specify the type of program being executed in the PROGRAM TYPE control and whether or not to view the execution in the VIEW control.
6. Click on the SAVE COMMAND TO LIST button to add the specified user program to the list of programs to be executed for the current execution mode.

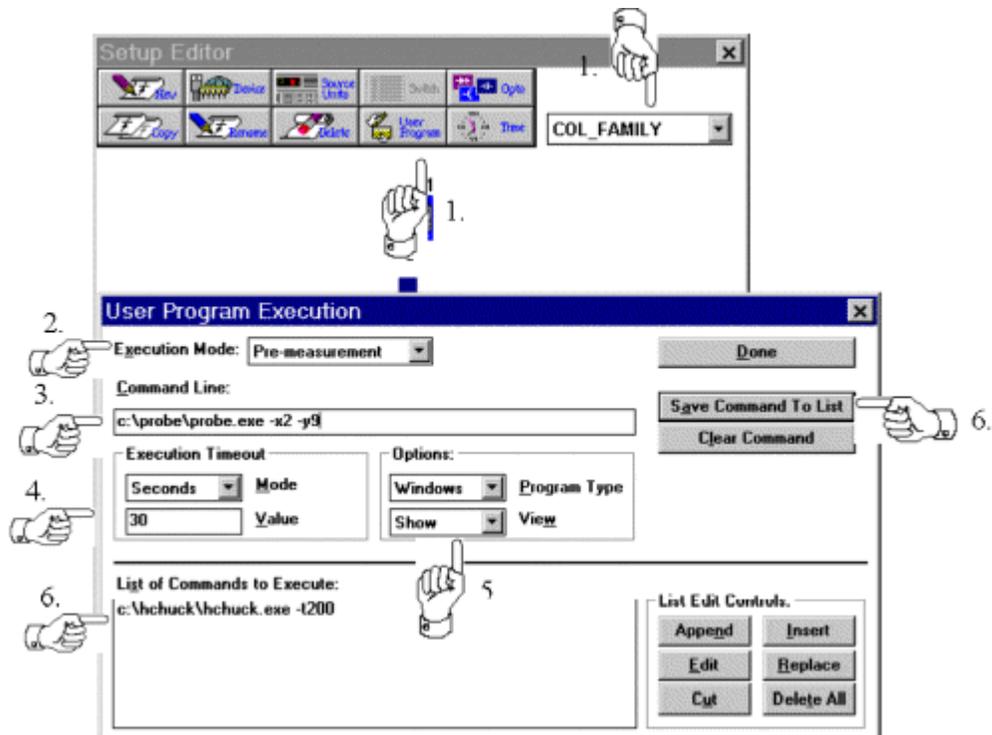


Figure 2-22: How to Define a User Program for Execution.

HOW TO EDIT A USER PROGRAM FOR EXECUTION:

1. From the Setup Editor setup window, select the USER PROGRAM button to display the User Program Execution dialogue box.
2. Specify whether the user program exists in the pre-measurement or post-measurement User Program execution list from the EXECUTION MODE control.
3. Highlight the command to be edited in the LIST OF COMMAND TO EXECUTE control.
4. Click on the EDIT button to place the user programs parameters in the definition controls.
5. Edit the user program parameters in the same manner as the user program was defined.

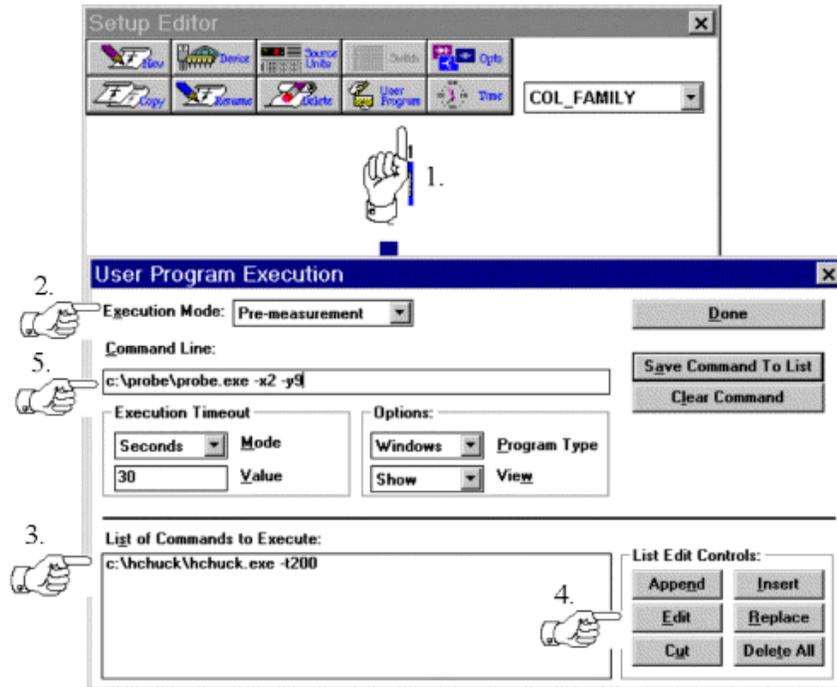


Figure 2-23: How to Edit User Program Execution.

USER PROGRAM EXECUTION NOTES



Communication with Win4145 from an external program must be done through the METRICS.OUT file. If the file is not used timeout errors will occur when executing external programs.

COMMUNICATION BETWEEN WIN4145 AND EXTERNAL PROGRAMS

Programs created for execution from within Win4145 must communicate with Win4145 through a file named METRICS.OUT in the Win4145 directory. Win4145 erases the METRICS.OUT file prior to the execution of an external program. After starting the

external program for execution Win4145 watches for the creation of the METRICS.OUT file and the return code placed in the file from the external program.

The METRICS.OUT file must be created by the external program when it has completed execution. Also, a return code must be written to the METRICS.OUT file by the external program. An ASCII 0 (zero) written to the file designates that execution completed successfully. Any other ASCII number written to the file designates an error occurred and all execution by Win4145 will stop.

Win4145 erases the METRICS.OUT file after the execution of the Win4145 setup.

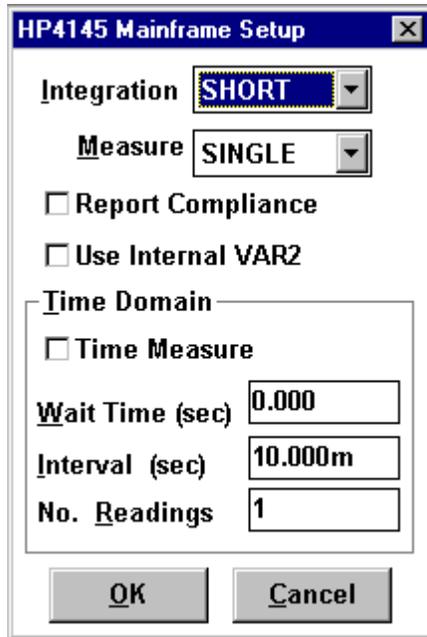
INSTRUMENT OPTIONS

The final button in the Setup Editor is used to set the instrument specific options available.

MAINFRAME OPTIONS

The Mainframe Setup dialogue box provides access to the HP4145 Integration and Time Domain controls. The Mainframe Setup dialogue box also includes the Single/Repeat Measure option along with the Report Compliance status switch. The latter features are unique to Win4145. The Single/Repeat Measure option and the Report Compliance status switch do not correspond to any of the front-panel instrument controls.

The configuration of the Mainframe Setup dialogue box can be uniquely specified for each test setup. Mainframe setup options are not specific to a single source unit; instead, mainframe setup options are applicable to every source unit in the test setup. In general, the default specifications in this dialogue box will be sufficient for most measurement applications. It will probably not be necessary for the user to configure the Mainframe Setup dialogue box each time a new test setup is created.



The Mainframe Setup dialogue box is opened by clicking the Setup Editor OPTIONS button after designating at least one Source Unit/DUT connection. The HP4145 Mainframe specifications are applicable to every source unit in the test setup. When a new test setup is created, the Mainframe Setup dialogue box can be designated without affecting the mainframe setup configuration in the original test setup(s).



The HP4145 Mainframe Setup dialogue box is opened by clicking the Setup Editor OPTIONS button after designating at least one Source Unit/DUT connection.

INTEGRATION TIME

In order to minimize the effects of line frequency interference and other noise sources, the HP4145 is equipped with three digital integration modes

1. **Short:** Digital integration is not performed.
2. **Medium:** The measurement result is the average of 16 samples taken during one line frequency period.
3. **Long:** The measurement result is the average of 256 samples taken during sixteen line frequency periods.

If your measurement data includes random variations in magnitude, consider selecting a longer integration time. To select a new integration time, click the scroll arrow at the right of the Integration field to display a list of available designations. Click on the desired option.

NOTE: As a default, the Integration field will be configured in SHORT mode. Selecting a longer integration time will increase the time required to complete the measurement.

SINGLE/REPEAT MEASURE MODE

SINGLE MEASURE

When Win4145 is configured in Single Measure mode, Win4145 will always configure the HP4145 Channel Definition Page and Source Unit Setup Page prior to initiating the test setup measurement. Win4145 will not pay attention to whether the instrument setup information is already loaded in instrument memory. As a default, Win4145 will operate in Single Measure Mode.

REPEAT MEASURE

The functionality of the Repeat Measure option is not equivalent to the functionality of the HP4145 front-panel Repeat Measurement key. The Repeat Measure option is exclusive to the Win4145 HP4145 Instrument Driver. This instrument enhancement is designed to improve measurement speed during the consecutively repeated execution of a single test setup.

Repeat mode eliminates the time spent redundantly configuring the instrument when the same test setup is repeatedly executed. When a test setup is executed in Repeat mode, Win4145 will determine if the present test setup information is already configured in instrument memory. This is determined by identifying whether the presently executed test setup is identical to the previously executed test setup. Win4145 determines if this is the case by tracking whether the user 1) changed focus to a different test setup, or 2) edited the test setup definition. If neither of these situations occurred, Win4145 will discern that the present test setup execution is a repetition of the previously executed test setup.

When Win4145 detects the repeated execution of a test setup, Win4145 will skip the instrument setup steps and immediately trigger the instrument to initiate the measurement. (An exception occurs during test setups that include a source unit configured in STEP mode. In terms of the HP4145, source units configured in STEP mode are applied to the DUT by repeated applications of a single stepped VAR2 source. When a test setup of this type is executed in Repeat mode, the Source Setup Page will be displayed momentarily while Win4145 updates the Start and Step value for the VAR2 source channel. The time required to update the Source Unit Setup page is minimal since only two fields require revision.) This will be apparent by the appearance of the List Display Page on the instrument's front panel almost immediately after the toolbar MEASURE button is selected. Win4145 will not spend time configuring the HP4145 Channel Definition Page and Source Setup Page. Configuring these HP4145 screens during a repeated

measurement is redundant because this information is still in instrument memory as a result of the last test setup execution.

The first time a test setup is executed in Repeat mode, Win4145 must send the test setup information to the instrument just as Win4145 does in Single mode. As long as the user repeatedly executes the same test setup, Win4145 will skip the instrument setup steps and immediately trigger the instrument to initiate the measurement each time the toolbar MEASURE button is selected. If the cycle is interrupted by executing a different test setup or editing the present test setup, the instrument configuration stored in instrument memory will no longer correspond to the active test setup. If the user initiates a new cycle of repeated measurements using the Repeat mode test setup, Win4145 will configure the instrument prior to the first measurement but eliminate the setup steps for each consecutively repeated execution of the test setup.

REPORT COMPLIANCE

Selecting the Report Compliance switch will configure Win4145 to report compliance events. If the Report Compliance switch is selected, Win4145 will halt the active measurement and display a message box when a compliance event is detected. The message box will identify the source unit in compliance and present the user with the options of retrying the test setup, aborting the test setup, or ignoring the message. After selecting an option, Win4145 will close the message box and proceed as instructed.

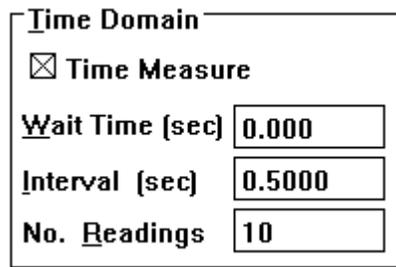


Figure 2-24: Win4145 Reports Compliance Events if the Report Compliance Switch is Selected in the Mainframe Setup Dialogue Box.

As a default, the Report Compliance switch is OFF. If you wish Win4145 to identify compliance events, make certain that the Report Compliance switch is selected in the Mainframe Setup dialogue box.

TIME DOMAIN

The HP4145 firmware includes a time domain utility that will return measured device parameters as a function of time. The HP4145 Time Domain utility is internal to the instrument and is distinct from the time domain utility provided in the Win4145 Setup Editor. The HP4145 Time Domain mode is configured with the Time Domain controls located in the HP4145 Mainframe Setup dialogue box. The Win4145 Time Domain utility is configured with the controls located in the Setup Time dialogue box opened by selecting the Setup Editor TIME button. For more information about configuring an HP4145 test setup using the Win4145 Time Domain utility, refer to *Chapter 2: The Setup Editor, Creating a Time Domain Test Setup*.



The screenshot shows a dialog box titled "Time Domain". It contains a checked checkbox labeled "Time Measure". Below this are three input fields: "Wait Time (sec)" with the value "0.000", "Interval (sec)" with the value "0.5000", and "No. Readings" with the value "10".

Figure 2-25: The HP4145 Time Domain Utility is Internal to the Instrument and is Distinct from the Win4145 Time Domain Utility.

FUNCTIONALITY

When the HP4145 Time Domain utility is implemented in a test setup, the VAR1 signal sweep is replaced with a time sweep. A family of curves measured with respect to time can be generated by incrementing the output of one source unit while the remaining source unit(s) output a constant signal. The source unit must be incremented by configuring the source unit in STEP mode (VAR2 source). All other source units connected to the DUT must

be configured in CONSTANT mode. (The STEP source is optional. The HP4145 Time Domain utility is functional when all of the source units connected to the DUT are configured in CONSTANT mode.) When the test setup is executed, TIME will be returned to the first column of the data window spreadsheet. Additional spreadsheet columns will be occupied by returned data. (The structure of the data window spreadsheet suggests that time be plotted as the independent variable, though any data vector can be designated as the independent variable from the Setup Data dialogue box.) If the user configures an HP4145 Time Domain test setup and later configures one of the source units in either of the SWEEP modes, the time domain utility will be disabled automatically.

TIME MEASURE

The Time Measure switch turns ON or OFF the HP4145 Time Domain utility. The HP4145 Time Domain utility supports test setups incorporating either all constant sources or constant sources with one stepped source. If one of the source units is configured in either of the SWEEP modes after the HP4145 Time Domain utility is selected, the HP4145 Time Domain utility will be disabled automatically.

WAIT TIME

The wait time is the amount of time the instrument will wait before initiating the source unit outputs. Wait time must be between 0 and 100 seconds.

INTERVAL

The interval is the amount of time the instrument will wait between consecutive measurements. The interval must be between 10 milliseconds and 10 seconds.

NO. READINGS

The No. Readings specification defines the total number of data points comprising the time sweep. If a source unit is configured in STEP mode, the No. Readings specification is the number of measured points that will be collected at each step value. A total of 500 or 1024 data points can be specified for the HP4145A or HP4145B respectively.

THE SOURCE UNIT SETUP DIALOGUE BOX

The setup conditions of each instrument source unit are independently controlled from a Source Unit Setup dialogue box. There is a Source Unit Setup dialogue box that corresponds to each HP4145 source unit. Each Source Unit Setup dialogue box is accessed from the Setup Editor. An example of a Source Unit Setup dialogue box is shown in Figure 2-26.

HOW TO DISPLAY A SOURCE UNIT SETUP DIALOGUE BOX:

In order to display a Source Unit Setup dialogue box, the corresponding source must be attached to the device schematic in the Setup Editor.

1. If only one source unit connection is designated at the same schematic pin, click once on the instrument icon to open the corresponding Source Unit Setup dialogue box.
2. If more than one source unit connection is designated at the same schematic pin, click once on the instrument icon to display a list of designated source unit connections.
3. Click once on the desired source unit designation to select the source unit; double-click on the source unit designation to open the corresponding Source Unit Setup dialog box.

The layout of a Source Unit Setup dialogue box is similar to the example pictured below.

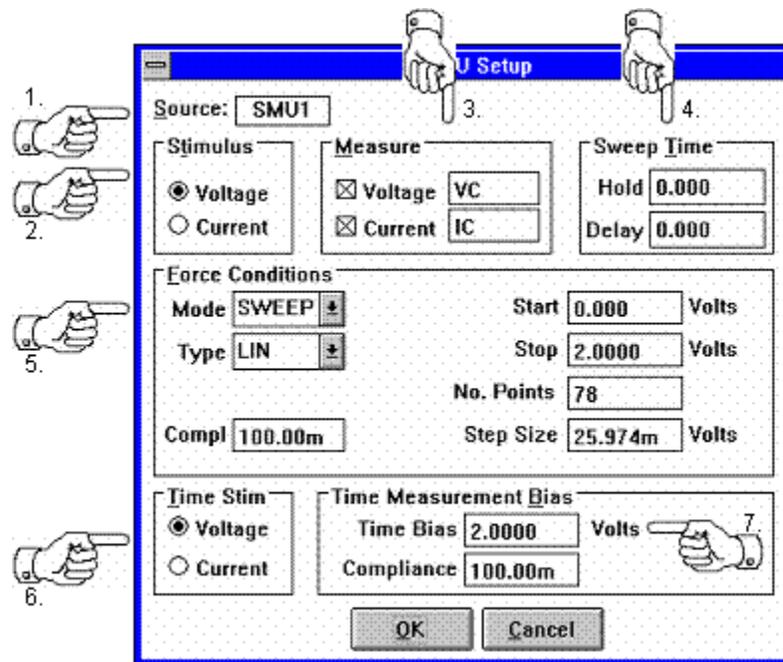


Figure 2-26: The Source Unit Setup Dialogue Box Corresponding to "SMU1" (Channel #1).

1. **Source Unit Identity Field:** This field is a static display identifying the suffix of the source unit label as designated in the Source Units dialogue box.
2. **Stimulus Controls:** This group consists of a pair of switches used to identify whether the forcing signal is a voltage or a current.
3. **Measure Controls:** This group includes the switches necessary to specify the type of signal the source unit will return as a measurement. This group also includes the necessary fields to label the returned measurements.
4. **Sweep Timing Controls:** This group includes the fields necessary to specify the Hold Time and Delay Time parameters.

5. Force Conditions Controls: The fields included within this group are used to define the shape of the forcing signal as well as the forcing signal's start and stop values. The compliance field is also included within this group.

6. Time Stimulus Switch: The Time Stimulus Switch allows the user to select the type of stimulus to use during Win4145 Time measurements.

7. Time Measurement Bias: The Time Measurement Bias value indicates the source value to be used during Win4145 Time measurements.

FORCE CONDITIONS CONTROLS

The Force Condition controls are used to specify the shape and boundaries of the forcing signal, as well as the forcing signal's compliance limit. Integration controls are configured from the HP4145 Mainframe Setup dialogue box.

MODE

The shape of the forcing signal is selected from a list of available signal outputs in the MODE field. The forcing signal may be characterized as a SWEEP, SYNCHRONIZED SWEEP, STEP, or CONSTANT supply. (These designations are the Win4145 equivalent of the VAR1, VAR1', VAR2, and CONST front panel source functions.) To select the desired sweep mode, click the MODE field scroll arrow. Clicking on the scroll arrow will display a list of available sweep modes. Click on the desired selection.

SWEEP MODE

Force Conditions	
Mode: SWEEP	Start: 0.000 Volts
Type: LIN	Stop: 2.000 Volts
Compl: 100.0m	No. Points: 201
	Step Size: 10.00m Volts



The sweep mode generates either a linear or logarithmic staircase sweep signal between two specified boundary values.

When creating a test setup that includes a second, synchronized sweep signal, the sweep application described in this section is the primary sweep signal.

HOW TO SOURCE A SWEEP SIGNAL

1. Select the "SWEEP" designation from the available options listed in the MODE field.
2. The data point distribution of the sweep signal can be either linearly or logarithmically distributed between the START and STOP values specified in Step #3. Select between a linear or logarithmic distribution by selecting the appropriate designation in the TYPE field.
3. Specify the range of the sweep signal in the START and STOP fields.
4. Hitting the tab key from the STOP field will move the cursor to the NO. POINTS field. Enter the number of data points that will comprise the sweep signal.
5. If a linear sweep type is selected in Step #2, the Sweep controls will include a STEP SIZE field. The STEP SIZE field will be calculated automatically after a value is entered in the NO. POINTS field and the cursor is moved to another location (or OK is selected). If desired, the calculated STEP SIZE can be updated by the user. If a new value is entered in the STEP SIZE field, the STOP field will be updated to accommodate the new STEP SIZE value. If any of the other fields are edited, the STEP SIZE field will be automatically updated to accommodate the change.

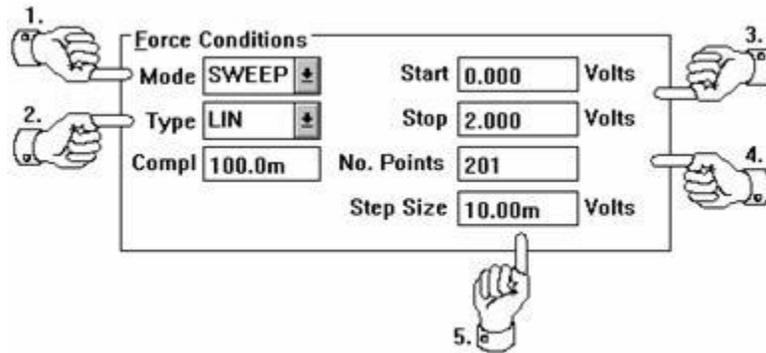
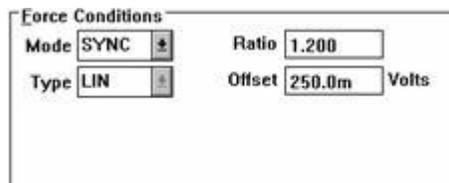


Figure 2-27: How to Source a Sweep Signal

SYNCHRONIZED SWEEP MODE



A synchronized sweep is a linear sweep signal that is synchronized in time with a primary sweep signal applied by another source unit.

A source unit can force a synchronized output only when a primary sweep signal is forced by another source unit in the test setup. If a synchronized sweep mode is designated in a test setup that does not include a primary sweep signal, Win4145 will display an error message when the user attempts to execute the test setup.

The data point distribution of both the primary sweep signal and the synchronized sweep signal must be linearly calculated. Win4145 will generate an error message if the user attempts to configure a synchronized sweep output in a test setup that includes a *logarithmic* primary sweep source. The data point distribution of the synchronized sweep signal is automatically designated as LINEAR. When a SYNC output is selected in the Force Conditions MODE field, the TYPE field is locked in the LINEAR configuration.

The stimulus of the synchronized sweep signal must match the stimulus of the primary sweep. In other words, if the primary sweep signal forces a voltage, the synchronized sweep signal must also force a voltage. If the synchronous signal is configured after the primary signal, the corresponding Stimulus switch will be selected automatically in the SYNC source. The alternative Stimulus switch will be unavailable to the user as indicated by the gray color of the switch.

While the *timing* of the primary and secondary sweep signals are synchronized, the magnitudes can differ. The magnitude of the synchronous signal is specified relative to the primary sweep by designating a constant ratio and/or a constant offset. The meaning of "ratio" and "offset" is explained in Figure 2-28.

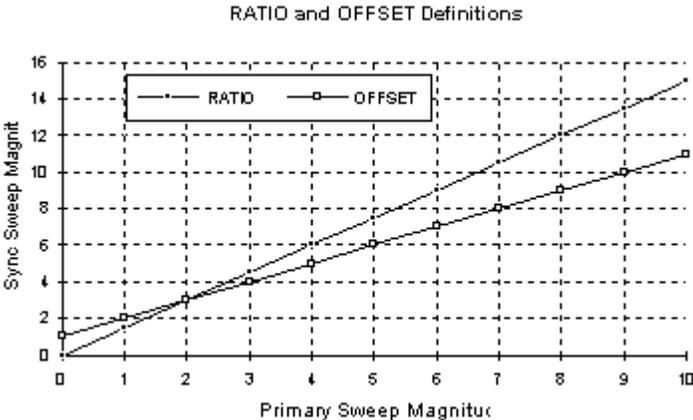


Figure 2-28: Synchronous Sweep "Ratio" and "Offset" Definitions.

When a RATIO is specified:

$$\text{Sync Sweep Mag} = \text{Primary Mag} * (\text{Ratio})$$

When an OFFSET is specified:

$$\text{Sync Sweep Mag} = \text{Primary Mag} + (\text{Offset}).$$

HOW TO SOURCE A SYNCHRONIZED SWEEP SIGNAL:

A synchronized sweep signal is functional only when a linear sweep signal is forced by another source unit.

1. Select the "SYNC" designation from the available options listed in the MODE field.
2. The data point distribution of both the primary and synchronous sweep signals must be linearly calculated. For this reason, the TYPE field will be locked in the LINEAR configuration.
3. The output stimulus of the synchronous sweep must match the output stimulus of the primary sweep. If the synchronous sweep is configured after the primary sweep is defined, the corresponding Stimulus switch (VOLTAGE or CURRENT) will be selected automatically. The alternative switch will be unavailable.
4. If desired, specify a value in the OFFSET field. When an offset value is specified, the offset component of the synchronized sweep magnitude is the sum of the primary sweep magnitude and the OFFSET value.
5. If desired, specify a value in the RATIO field. When a ratio value is specified, the ratio component of the synchronized sweep magnitude is the product of the primary sweep magnitude and the RATIO value. A synchronized ratio can be specified in combination with a synchronized offset.

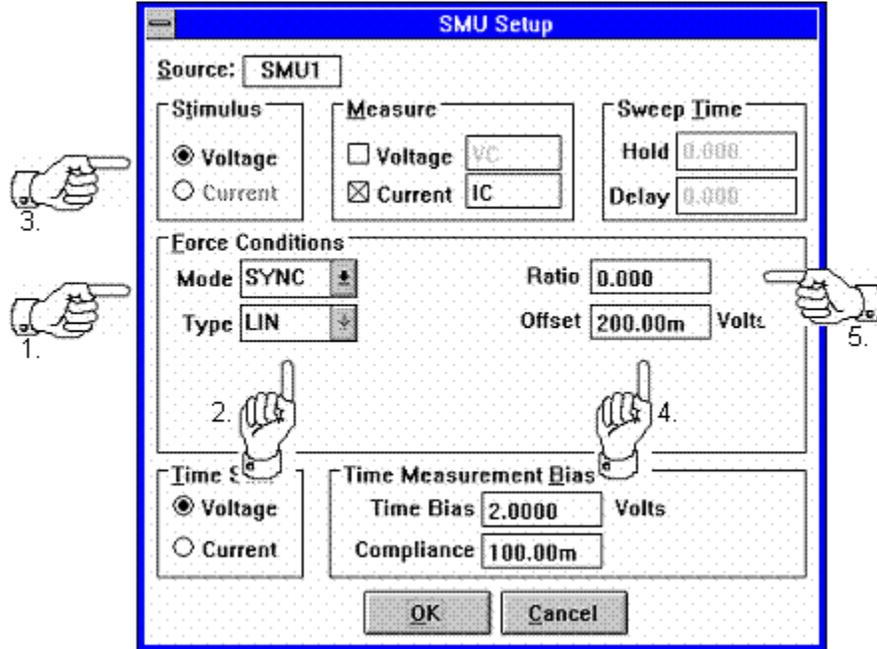


Figure 2-29: How to Source a Synchronized Sweep Signal

STEP MODE

Force Conditions				
Mode	STEP	Start	10.00u	Amps
Type	LIN	Stop	60.00u	Amps
Compl	1.000	No. Steps	6	
		Step Size	10.00u	Amps

The step mode generates a constant output signal while a second source unit generates a sweep signal. After the sweep signal reaches the specified STOP value, the magnitude of the step output is incremented by the STEP value and the sweep output is triggered again. This process continues until the STOP value of the step signal is reached.

A STEP mode output signal can only be incremented by a constant value. When "STEP" is designated in the Force Conditions MODE field, the TYPE field will be locked automatically in the LINEAR configuration.

HOW TO SOURCE A STEP SIGNAL

1. Select the "STEP" designation from the available options listed in the MODE field.
2. The STEP mode output signal can only be incremented by a constant value. For this reason, the TYPE field is locked in the LINEAR configuration.
3. Specify the range of the step signal in the START and STOP fields.
4. Specify the increment quantity in the NO. STEPS field.
5. The increment size will be displayed in the STEP SIZE field. After a value is entered in the NO. STEPS field and the cursor is moved to another location (or OK is selected), the STEP SIZE field will be calculated automatically. If desired, the calculated STEP SIZE can be updated by the user. If a new value is entered in the STEP SIZE field, the STOP field will be updated to accommodate the new STEP SIZE value. If any of the other fields are edited, the STEP SIZE field will be updated automatically to accommodate the change. The step mode can only be used in test setups that also include the specification of a sweep signal. A source unit configured in a step mode will force a constant value signal while a second source unit generates a sweep signal. After the sweep signal has been applied, the step value will be incremented in preparation for another sweep application. This process continues until the stop value of the step signal is reached.

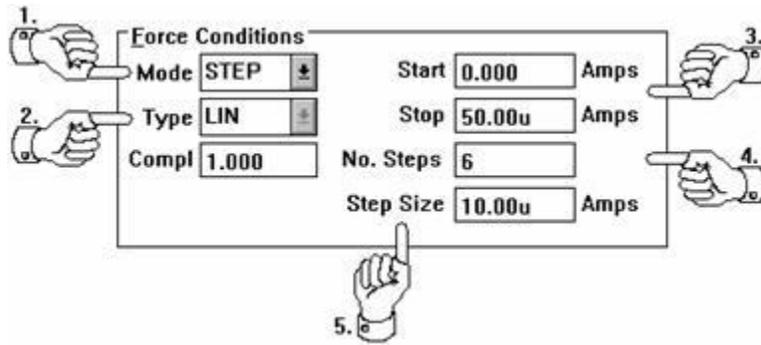
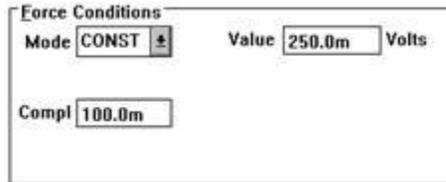


Figure 2-30: How to Source a Step Signal

CONSTANT MODE



HOW TO SOURCE A CONSTANT SIGNAL

1. Select the "CONST" designation from the available options listed in the MODE field.
2. Specify the signal magnitude in the VALUE field.

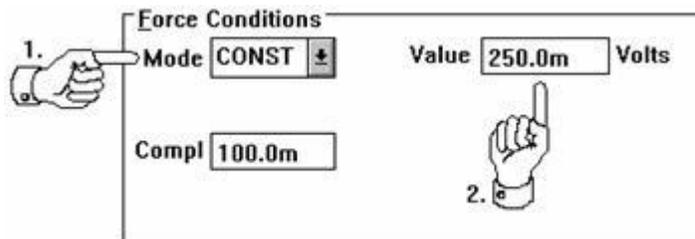


Figure 2-31: How to Force a Constant Signal

TYPE

The TYPE field is used to designate whether the data point distribution of a sweep output is linearly or logarithmically calculated. This field is displayed when the SWEEP designation is selected in the MODE field. If the STEP or SYNC mode is active, the TYPE field is locked in the LINEAR configuration.

START

The START field is used to specify the starting value of the sourcing signal. This field is displayed when the SWEEP or STEP designation is selected in the MODE field.

STOP

The STOP field is used to specify the stopping value of the sourcing signal. This field is displayed when the SWEEP or STEP designation is selected in the MODE field.

NO. POINTS

The NO. POINTS field is used to specify the data point quantity in a linear sweep signal. This field is displayed when the SWEEP designation is selected in the MODE field and the LINEAR designation is selected in the TYPE field.

PTS/DEC

The PTS/DEC field reports the number of data points that will be measured per sweep decade. The PTS/DEC field can not be edited by the user. The number reported in this field is determined by the logarithmic selection configured in the TYPE field.

NO. STEPS

The NO. STEPS field is used to specify the increment quantity of a step signal. This field is displayed when the STEP designation is selected in the MODE field.

STEP SIZE

The STEP SIZE field designates the linear increment of a sweep or step signal. The value of this field is automatically calculated after the user specifies either the NO. POINTS in a linear sweep or the NO. STEPS in a step output. If the value of this field is changed by the user, the value of the STOP field will be updated to accommodate the change. This field is displayed when the STEP designation is selected in the MODE field. The STEP SIZE field is also displayed when the SWEEP designation is selected in the MODE field, and the LINEAR characteristic is selected in the TYPE field.

VALUE

The VALUE field is used to specify the magnitude of a constant signal. This field is displayed when the CONSTANT designation is selected in the MODE field.

OFFSET AND RATIO

These fields are used to configure the output a synchronous output. These fields are displayed when the SYNC designation is selected in the MODE field.

COMPLIANCE

The COMPLIANCE field is used to specify the limiting magnitude of a measured signal. When a source unit is forcing voltage and measuring current, a current compliance must be specified. Similarly, if a source unit is sourcing current and measuring voltage, a voltage compliance must be specified. Depending upon the source units being used, the compliance limit may be locked at a particular value.

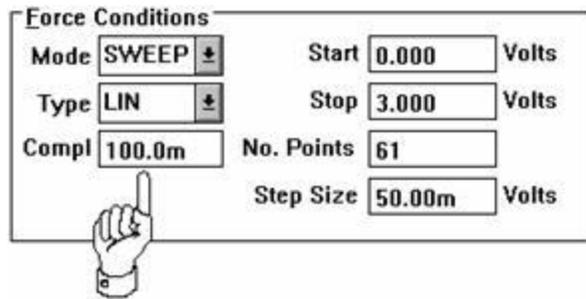


Figure 2-32: Designating a Measurement Compliance

The compliance limit allows the user to execute a test setup that is within an acceptable power range of the device by limiting the operating range of the source unit. For example, if a 100mA compliance limit is specified for a source unit that is sourcing a sweep voltage and returning a current, the source unit will supply an increasing voltage signal until a 100mA current is returned.

The maximum compliance limit that Win4145 will allow is determined by the operating boundaries of the instrument. Please refer to Chapter 3 of the HP4145A/B Operations Manual for an overview of the HP4145 compliance configurations and power limitations.

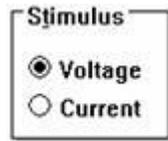
DETECTING COMPLIANCE EVENTS

If the Report Compliance switch is selected in the Mainframe Setup dialogue box, Win4145 will halt the active measurement and display a message if a compliance limit is detected. (The Mainframe Setup dialogue box is opened by clicking the Setup Editor OPTIONS button.) The message box will identify the source unit in compliance and present the user with the options of retrying the test setup, aborting the test setup, or ignoring the message. After selecting an option, Win4145 will close the message box and proceed as instructed.

As a default, the Report Compliance switch is OFF. If you want Win4145 to identify compliance events, make certain that the

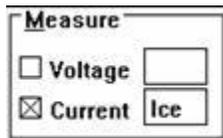
Report Compliance switch is selected in the Mainframe Setup dialogue box.

STIMULUS CONTROLS



Depending upon the source, one of the two switches may be unavailable. For example, the HP4145 Voltage Monitor Units (VMs) are designed for high resolution voltage measurements. VMs can NOT be configured to force a voltage.

MEASURE CONTROLS



SPECIFYING RETURNED DATA

Depending upon the particular HP4145 source, Win4145 may be configured to return the forcing signal, the forcing signal complement, both, or none. For an overview of each source's capability, please refer to Chapter 3 of the HP4145A/B Operations Manual.

A source's measurement configuration is specified by turning on or off the VOLTAGE and CURRENT switches. Depending upon the particular source, one of the two switches may be unavailable.

HOW TO SPECIFY RETURNED DATA:

1. Open the Source Unit Setup dialogue box corresponding to the source that will return data.
2. Click on one or both of the measure switches.
3. After you select either of the measure switches, a default text label will appear in the field next to the corresponding switch. If the default label is unacceptable, specify an alternative name for the measured characteristic.

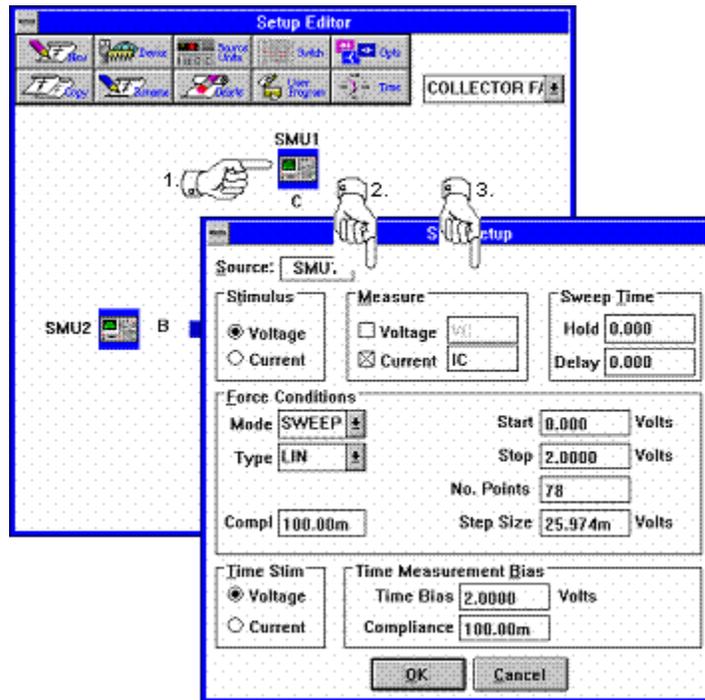


Figure 2-33: How to Specify Returned Data

LABELING MEASURED DATA

All of the data that corresponds to a single curve is collectively referred to as a "data vector". Each data vector is identified by a "data vector label" that must be defined in the text field to the right of the VOLTAGE or CURRENT switch. A data vector label can be any alphanumeric string up to three characters in length.

The presence of only two data vector fields does not mean that only two data vectors can be specified in a test setup. In fact, by using a combination of step and sweep sources, up to 42 data vectors per test setup can be measured.

Test setups that use a combination of step sources and sweep sources can return sequential and non-sequential data vectors. Test setups that use a combination of constant sources and sweep sources can only return non-sequential data vectors.

SEQUENTIAL DATA VECTORS

A sequential data vector is a device characteristic that is described by a family of unique curves. Each curve is measured in response to a stepped bias condition. Sequential data vectors are the result of test setups that include stepped sources and repeated sweeps.

For example, consider the DC collector characteristics of a bipolar transistor. The DC collector characteristics were obtained by generating a family of ICE vs. VCE curves. This setup was created by applying a stepped current supply to the base and a swept voltage supply to the collector. The source unit connected to the collector returned both collector voltage (VCE) and collector current (ICE). Each time the base current was stepped to a new value, a unique ICE curve was obtained in response to the collector voltage sweep. The result of this test setup was a family of unique ICE curves. ICE is a sequential data vector because collector current is described by a family of unique curves measured in response to stepped base current.

NON-SEQUENTIAL DATA VECTORS

A non-sequential data vector is a device characteristic that is described by a single curve.

For example, consider the example presented in the previous section, *Sequential Data Vectors*. The DC collector characteristics of a bipolar transistor were obtained by generating a family of ICE vs. VCE curves. This setup was created by applying a stepped current supply to the base and a swept voltage supply to the collector. The source unit connected to the collector returned both collector voltage (VCE) and collector current (ICE). Because the base current was stepped, the test setup returned a family of ICE curves, but the voltage sweep applied to the collector during each base current step was the same. Therefore, VCE is a non-sequential data vector because VCE can be described by a single curve.

SWEEP TIMING CONTROLS

Sweep Time	
Hold	5.000m
Delay	5.000u



A hold time and delay time may be specified when forcing a sweep signal. Hold time is the amount of time the instrument will wait before applying the sweep output. Delay time is the time the instrument will wait between consecutively forced sweep points.

CHAPTER 3: THE FILE MANAGER

WHAT IS A PROJECT FILE?

A project file is a collection of one or more test setups stored as a single unit. A project file provides the capability to store a group of test setups that share a common association.

WHAT A PROJECT FILE INCLUDES

5. A project file includes the associated test setup(s).
6. A project file includes any of the measured data and plots corresponding to the test setups.
7. A project file includes the Instrument Driver selection required to execute the test setup(s).



When a project file is opened, the necessary instrument driver is automatically connected to Win4145. Any of the existing test setups can be executed immediately.

OPENING A PROJECT FILE

Only one project file can be opened at a time. Project files are opened by selecting the file name and directory specified when the project file was saved.

HOW TO OPEN A PROJECT FILE

1. Select FILE/OPEN from the File menu bar item. This will open the File Open dialogue box.
2. Select the directory the file was saved within and highlight the file name.
3. Click the OK button to open the specified project file and restore control to the desktop. All of the corresponding data window spreadsheets and plot windows will be minimized along the bottom of the desktop.

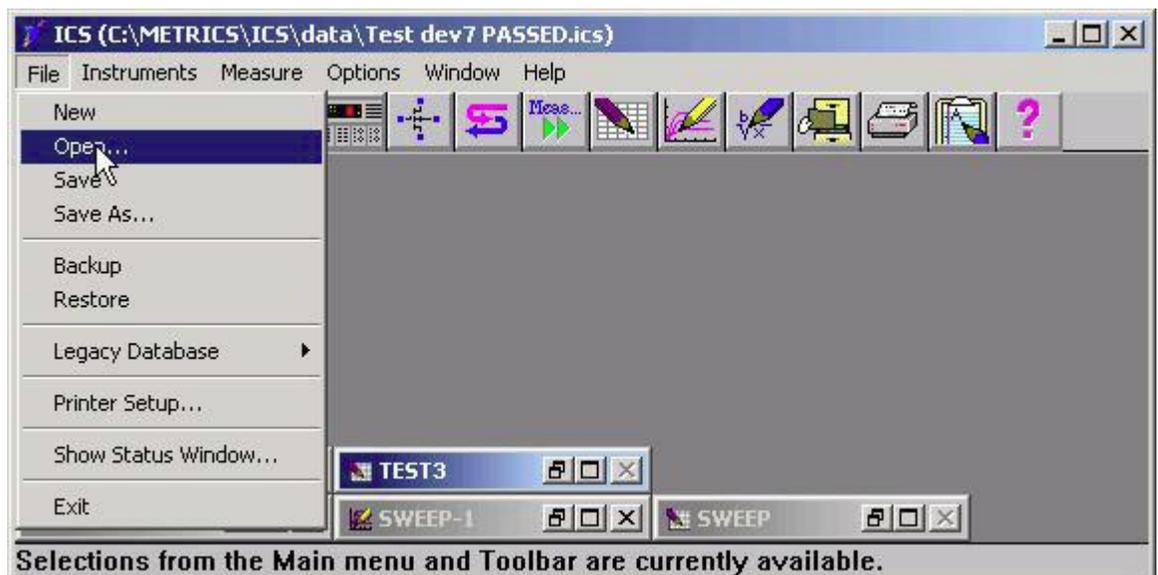


Figure 3-1: Opening an Win4145 Project File

SAVING A PROJECT FILE

Project files are saved by selecting the file name and directory specified when the project file is saved.

HOW TO SAVE A PROJECT FILE:

1. Select FILE/SAVE AS from the File menu bar item. This will open the File Save dialogue box.
2. Select the directory the file was saved within and highlight the file name.
3. Click the OK button to save the specified project file and restore control to the desktop. All of the corresponding data window spreadsheets and plot windows will be minimized along the bottom of the desktop.

DELETING A PROJECT FILE

Project files are deleted by selecting the file name and directory specified when the project file is saved.

HOW TO DELETE A PROJECT FILE:

1. Select FILE/DELETE AS from the File menu bar item. This will open the File Save dialogue box.
2. Select the directory the file was saved within and highlight the file name.
3. Click the OK button to save the specified project file and restore control to the desktop. All of the corresponding data window spreadsheets and plot windows will be removed from the bottom of the desktop.

MEASUREMENT TYPES

Win4145 can be configured to execute single, repeat, append, sequential, or manual types of measurements. The measurement type is selected from the SINGLE, REPEAT, APPEND, and AUTO SEQ buttons on the Measurement Remote Control dialogue box.



Figure 4-1: Measurement Types Available in Win4145.

SINGLE MEASUREMENT TYPE

Single measurement executes the current setup one time and updates the data and plot windows with the data returned from the driver for the measurement.

REPEAT MEASUREMENT TYPE

Repeat measurement executes the current setup continuously and updates the data and plot windows with the data returned from the driver for the measurement until the Stop button is pressed.

APPEND MEASUREMENT TYPE

Append measurement executes the current setup one time and appends the data from the current measurement to the data from previous measurements. The data and plot windows are then updated with the old and appended data.

AUTO SEQUENCE MEASUREMENT TYPE

Auto Sequence measurement executes each of the setups in the sequence or setups defined in the Measure Sequence dialogue one time and updates respective data and plot windows with the data returned from the driver for the measurements.

MEASUREMENT MODES

Win4145 can be configured to execute standard, bias delay, and time measure mode measurements. The measurement type is selected in the mode control, on the Measurement Remote Control dialogue box, and may be executed for Single, Repeat, Append, and Auto Seq measurement types.



Figure 4-2: Measurement Modes Available in Win4145.

STANDARD MEASUREMENT MODE

Executes the standard measurement portion of the setup as one complete measurement.

BIAS DELAY MEASUREMENT MODE

Executes the time measurement portion of the setup in a bias delay manner. *Refer to Creating a Time Domain Test Setup in Chapter 2 for a description of Bias Delay Measurements.*

TIME MEASURE MEASUREMENT MODE

Executes the time measurement portion of the setup in a time based measurement manner. *Refer to Creating a Time Domain Test Setup in Chapter 2 for a description of Time Measurement mode measurements.*

INTERACTIVE MEASUREMENT CONTROLS

Win4145 can interactively change Measurement Integration from the Measurement Remote control dialog by a press of the corresponding button.



Figure 4-3: Interactive Measurement Controls Available in Win4145.

The integration button is a fast way to change the measurement settings without going back to the setup editor. By default the button is set to *User* which then uses the Integration time setting selected in the **Instrument Options** of the *Setup Editor*.

Click the integration button to select between *Short*, *Medium*, *Long*, and *User*. The value selected by this button will take precedence over the settings in the *Setup Editor*.

EXECUTING A TEST SETUP

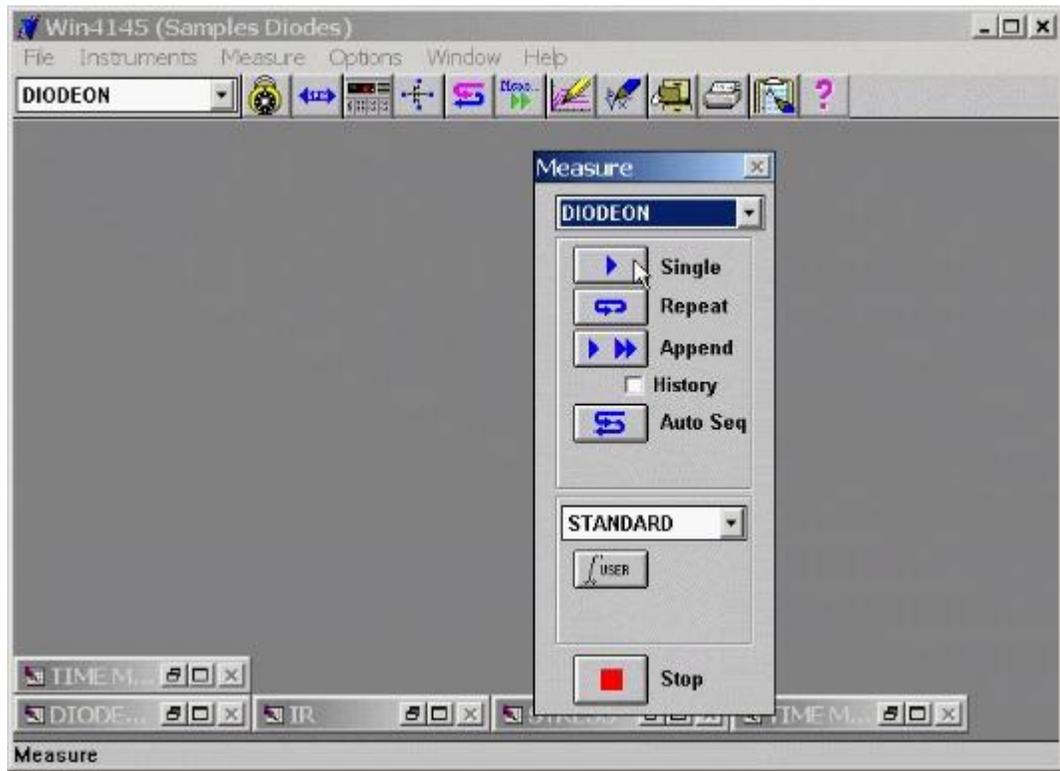


Figure 4-4: How to Execute a Test Setup.

HOW TO EXECUTE TEST SETUP:

1. Click the toolbar Measure button to display the Measurement Remote Control.
2. Select the active test setup from the Setup list combo box located at the top of the Measurement Remote Control.
3. Select the desired measurement mode from the measurement mode combo box.
4. Click on the measurement type button corresponding to the desired measurement type to be performed.

EXECUTING A SEQUENCE OF TEST SETUPS

Win4145 allows you to execute a sequence of project file test setups. To execute a sequence of test setups, define the test setup sequence in the Measure Sequence dialogue box and select the AUTO SEQ button on the Measurement Remote Control. When the AUTO SEQ button is clicked, Win4145 will execute the test setup sequence identified in the Measure Sequence dialogue box.

DEFINING A TEST SETUP SEQUENCE



The Measure Sequence dialogue box can be opened by selecting the toolbar Sequence button or by optionally selecting MEASURE/SELECT SEQUENCE from the measurement mode menu bar.

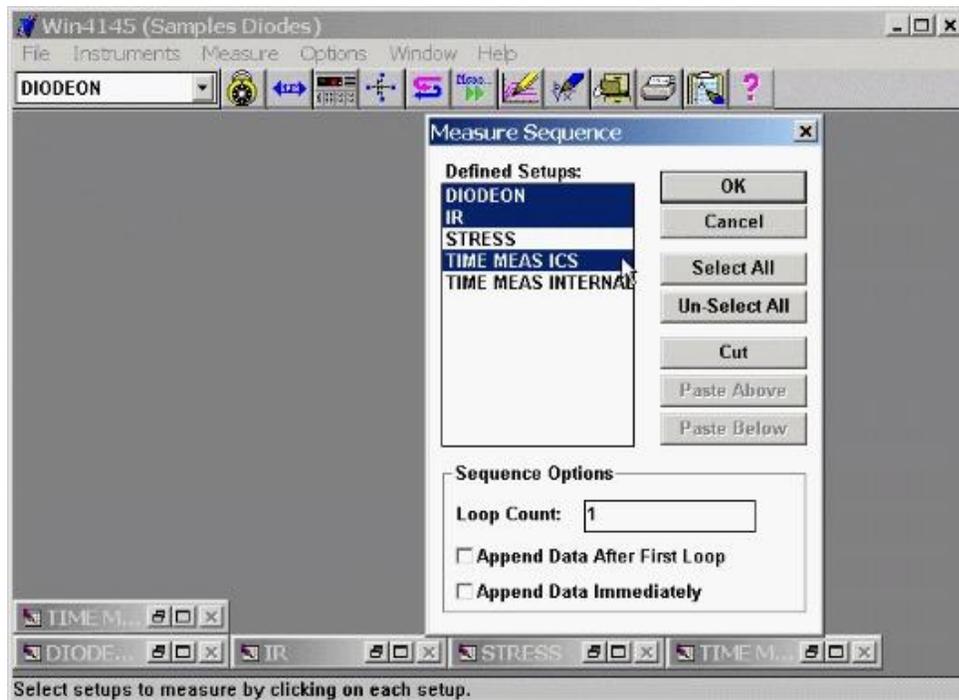


Figure 4-5: How to Define a Test Setup Sequence.

HOW TO DEFINE A TEST SETUP SEQUENCE:

1. Click the toolbar Sequence button to open the Measure Sequence dialogue box. The Measure Sequence dialogue box can also be opened by selecting MEASURE/SELECT SEQUENCE from the measurement mode menu bar.
2. All of the test setups included in the project file are displayed in the Defined Setups window. Define a test sequence by selecting the desired test setups. A test setup is selected by highlighting the setup name. Setup selections can be canceled by clicking on the setup name a second time. The setups do not have to be adjacent to each other in order to be selected.
3. The test sequence will be executed from the top down, not the order in which the setups were selected. To change the test setup order, select the test setup that will be repositioned. Make sure this is the only test setup highlighted. Click the CUT button. Highlight the test setup that will be either above or below the inserted test setup. Click the appropriate paste button, either PASTE ABOVE or PASTE BELOW, to insert the test setup. Make sure that the selected setups rely upon compatible test fixture hardware arrangements. Win4145 will not pause between setup executions to provide the opportunity to reconfigure device orientation or test fixture configuration.
4. After the desired test setups have been selected and ordered, click OK to close the Measure Sequence dialogue box and restore control to the Win4145 desktop.
5. The Sequence options allow the sequence to be looped for a specified number of loops. The append options allow the data from the sequence to be appended to data from the previous execution of the sequence.

EXECUTING THE TEST SEQUENCE



Figure 4-6: How to Execute Sequential Test Setups.

HOW TO EXECUTE SEQUENTIAL TEST SETUPS:

1. Click the toolbar Measure button to display the Measurement Remote Control.
2. Select the active test setup from the Setup list combo box located at the top of the Measurement Remote Control.
3. Select the STANDARD measurement mode from the measurement mode combo box.
4. Click on the AUTO SEQ measurement type button.

WHAT IS A DATA WINDOW?

	VB	IB
1	0.000	-8.9400p
2	-266.67m	-9.7400p
3	-533.33m	-13.160p
4	-800.00m	-17.500p
5	-1.0667	-18.680p
6	-1.3333	-549.70p
7	-1.6000	-15.240p
8	-1.8667	-490.00p
9	-2.1333	-560.18p
10	-2.4000	-550.46p

Figure 5-1: Example of a Data Window Spreadsheet.

Data window spreadsheets provide the capability to record and edit measurement results. Each data window spreadsheet corresponds to a specific test setup. Because each data window is organized in a spreadsheet structure, data can be edited by cell, row, or column.



A data window spreadsheet stores the measured and calculated data that corresponds to a single test setup. Each test setup is associated with a single data window spreadsheet.

The contents of a data window spreadsheet will be automatically updated each time the corresponding test setup is executed, and a data window spreadsheet can also be edited by the user. When a data window spreadsheet is updated by re-executing the test setup, the plot window will be updated automatically. If the data window spreadsheet is edited, the plot window can be updated upon user command.

As soon as a test setup is created, a corresponding data window spreadsheet by the same name will be created in minimized form along the bottom of the

Win4145 desktop. *Minimizing* a data window will reduce the window to an icon positioned along the base of the desktop.



A data window icon is restored to an open view by clicking twice on the window icon. Clicking once on a data window icon will open the window control menu. The window control menu can also be accessed from an open view. Click once on the window's system menu icon located to the far left of the title bar.

EXPLORING THE DATA MENU BAR

When a data window is active, the menu bar includes pull-down lists particular to data window functions. To display the data menu bar, open or select a data window as the active desktop window, or click once on a data window icon.

FILE

The FILE drop-down list is available in all three menu bars. It includes the file and print commands relevant throughout the entire application.

EDIT

The EDIT drop-down list includes enhanced editing features for manipulating data window spreadsheets. Selectable items include DELETE, CUT, COPY, and PASTE. Use CUT or COPY to copy the selected data to the clipboard. Use PASTE to paste clipboard data into the selected data. Use DELETE to delete the selected data.

ANALYSIS

The ANALYSIS drop-down list provides a command to invoke the Transform Editor. The Transform Editor provides the capability to add new data columns to a data window spreadsheet by defining new data vector expressions based upon the measured data vectors.

WINDOW

The WINDOW drop-down list is available in all three menu bars. This is a Windows feature that includes the standard window and desktop arrangement commands. The NEW PLOT and UNHIDE DATA WINDOW functions are also included in the WINDOW pull-down list.

HELP

The HELP drop-down list is also available in all three menu bars. The HELP menu selection is used to invoke the On-Line Help Program.

CREATING DATA WINDOW SPREADSHEETS

A corresponding data window spreadsheet will be automatically created each time a new test setup is created. After creating a new test setup, a data window icon will appear at the base of the desktop. All of the measured data for the recently created test setup will be written to this data window spreadsheet.



The link between the data window spreadsheet and the test setup is dynamic. The contents of the data window spreadsheet will be updated each time the test setup is executed.

Because of the link between the data window spreadsheet and the associated test setup, the data window spreadsheet will automatically have the same name as the test setup.

SETTING DATA PRECISION

The precision of the numerical data stored in a Win4145 project file is defined by the measurement precision of the instrument. The measurement precision of the instrument is often related to the specific setup configuration of the hardware. Each Win4145 instrument driver returns data at the maximum instrument precision possible for each specific measurement setup.



While the data precision recorded in a project file is fixed by the instrument and the specifics of the measurement setup, the precision of the numerical data displayed in a data window spreadsheet can be controlled by the user.

Data window spreadsheet precision is controlled from the Workspace dialogue box. The data window spreadsheet precision can be varied to display between 3 and 7 significant digits (inclusive). If the user designates a level of precision that exceeds the precision of the data returned by the instrument, data will be displayed at the measured precision.

Data window spreadsheet precision is a global specification. This means that the designated spreadsheet precision is recorded in the ICS35.INI file. Every data window will be created and displayed at the precision identified in the ICS35.INI file independent of project file.

HOW TO DESIGNATE SPREADSHEET PRECISION:

1. Select OPTIONS/WORKSPACE from the measurement mode menu bar. This will open the Workspace dialogue box.
2. The Workspace dialogue box includes a group of controls labeled "Display". The Display Group includes a Significant Digits field. Designate a value between "3" and "7" (inclusive).
3. Select the OK button to close the Workspace dialogue box and restore control to the desktop.
4. It is not necessary to restart Win4145. The new precision specification will be applied to the desktop immediately. If a data window spreadsheet was displayed while changing the precision, move the vertical scroll box or minimize and restore the window in order for the opened window to display the new precision.

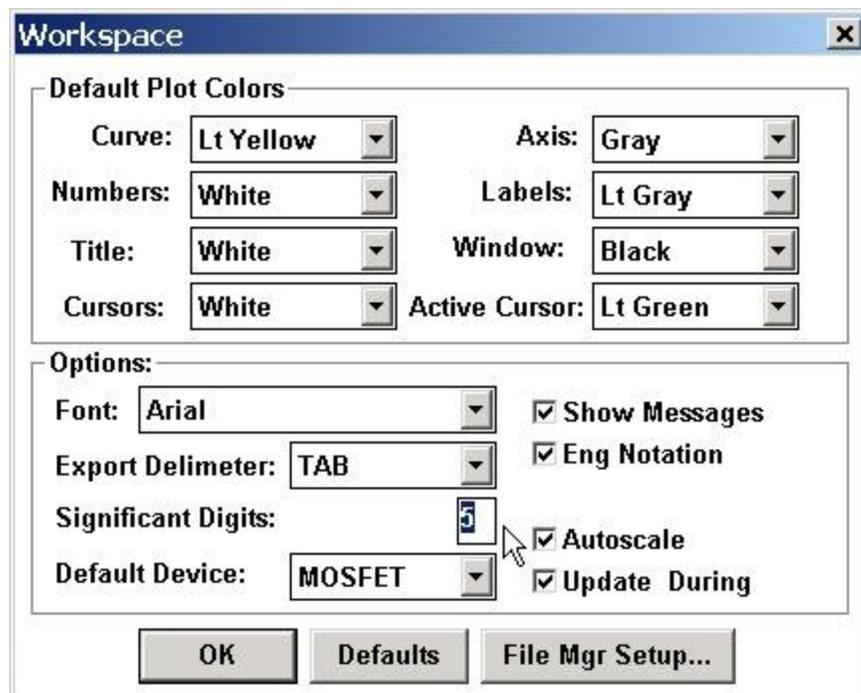


Figure 5-4: How to Designate Spreadsheet Precision

AUTO ANALYSIS WITH THE TRANSFORM EDITOR

The Transform Editor provides the user with the capability to define data vectors that are functions of measured data. These calculated data vectors are called "transforms". Transforms are analogous to what some instruments call "user functions". After one or more transform(s) are defined, each resultant data vector is appended to the next available column in the corresponding data window spreadsheet. There is no distinction between measured and calculated data vectors (transforms) from the perspective of a data window spreadsheet. Transforms can be manipulated and plotted in the same way as measured data vectors.



The Transform Editor provides the user with the capability to define data vectors that are functions of measured data. These calculated data vectors are called "transforms". Transforms are analogous to what some instruments call "user functions".

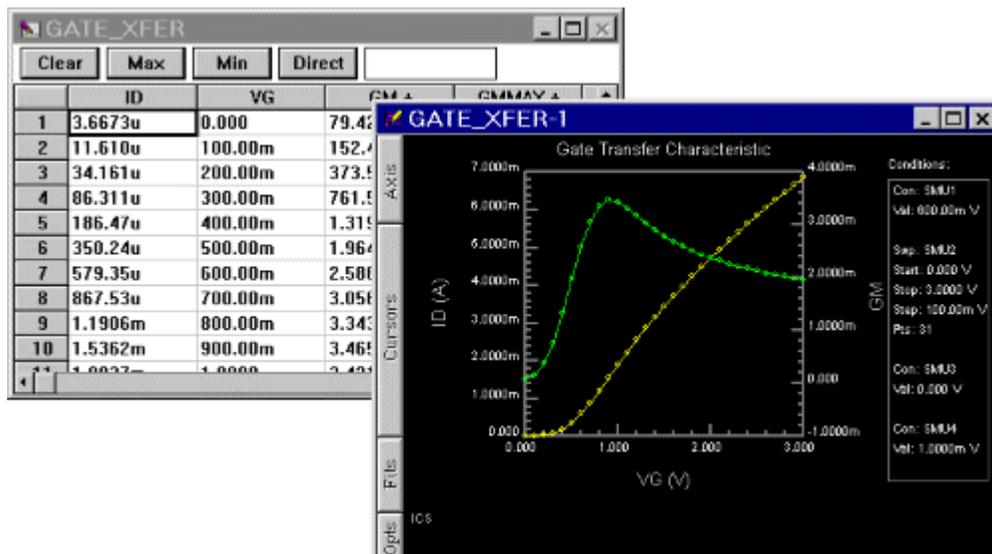


Figure 5-5: The Transform Editor provides the User with the Capability to Define Data Vectors that are Functions of Measured Data.



The Transform Editor will calculate the transforms as they are defined, therefore it is recommended that the setup be executed to gather data that the transforms will operate upon. Otherwise some data transforms will return errors when initially defined since the data does not yet exist.

THE TRANSFORM EDITOR DIALOGUE BOX

Transforms are algebraic equations written in the form VECTOR = EQUATION. VECTOR is the name of the resultant data vector that will be displayed in the data window spreadsheet. EQUATION is an algebraic expression based upon the measured or calculated data vectors.

The primary feature of the Transform Editor dialogue box is the Transform combo box. Below the Transform combo box are the fields, lists, and buttons used to define, edit, and save transform expressions. A detailed overview of the Transform Editor is shown below:

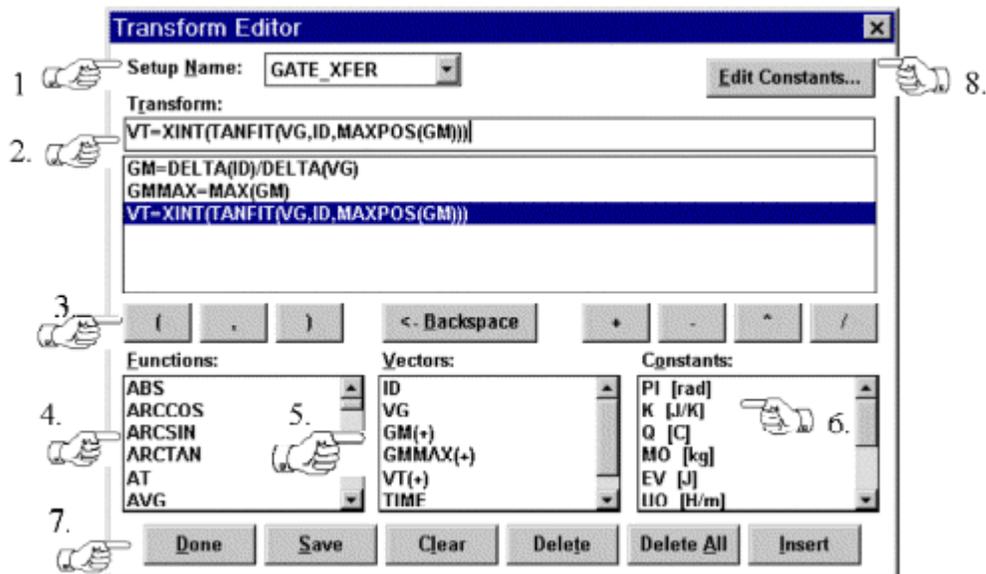


Figure 5-6: The Transform Editor dialogue box.

Setup Name: The Setup Name combo box is used to select the setup in which the transform will be defined.

Transform: The Transform edit field is used to define a transform in the form VECTOR = EQUATION. All transforms defined for the active test setup are displayed in the list box below the Transform edit field. Selecting, or double clicking on, a transform in the list box allows the user to edit or delete a transform that has already been defined in the active setup.

Character Buttons: The Character buttons include the entire strip of buttons located underneath the Transform combo box. The Parenthesis, Comma, Plus (+), Minus (-), Multiplication (*), and Division (/) buttons insert the respective character in the Transform edit field at the current carrot position. The "<-Backspace" button is equivalent to hitting the keyboard backspace key.

Functions List Box: The Functions list box displays a list of pre-defined math functions. A math function can be inserted into the active transform expression by double-clicking on the function name. When a math function is inserted into a transform expression, a left parenthesis will follow the function. This is intended to prompt the user for a function parameter such as a data vector or another transform vector.

Vectors List Box: The Vectors list box display a list of available data vector and transform vector names. A data or transform vector can be inserted into the active transform expression by double-clicking on the vector name.

Constants List Box: The Constants list box displays a list of pre-defined and user-defined constants. A constant can be inserted into the active transform expression by double-clicking on the constant name.

Transform Editing Push Buttons: The Transform Editing push buttons are located on the lower edge of the dialogue box. Their functionality is as follows:

- a. Done: Closes the Transform Editor dialogue box and restores control to the previously active window.
- b. Save: Saves the active transform to the list of defined transforms.
- c. Clear: Clears the Transform edit field.
- d. Delete: Deletes the active transform from the list of defined transforms.
- e. Delete All: Deletes all of the defined transforms from the active test setup.

f. Insert: Inserts the active transform to the list of defined transforms at a location before the transform highlighted in the list of transforms.

Edit Constants Button: The Edit Constants button displays the Constant Editor dialogue box. The Constant Editor dialog box allows the user to edit the values or names of existing constants as well as define new constants. For more information about editing or adding constants, please see *Editing Constants* later in this section.

Transform Editor Operation

Transforms are algebraic equations written in the form VECTOR = EQUATION. VECTOR is the name of the resultant data vector that will be displayed in the data window spreadsheet. EQUATION is an algebraic expression based upon the measured data vectors. The Transform Editor includes the standard math operators (*, /, +, and -), parenthesis, and a library of standard math functions and constants that can be used to write an equation.

Transforms also contain the ability to use a result of a previous transform as a parameter. This allows the user to "nest" or "cascade" transform expressions. The effect is the ability to write a complete extraction algorithm in a single transform using the Transform Editor.

Transforms also contain the ability to use any vector from any setup in the current project as a source vector. This allows the user to perform calculations based upon more than one measurement. The syntax is SETUPNAME|VECTOR (Setup Name | (Pipe character) Vector Name.

There is a limit to the number of transforms that can be created in a single data window spreadsheet (40, including data vectors). However, it is a good idea to keep the total number of data vectors within a single spreadsheet (measured and calculated) below 35.



The Transform Editor is displayed by clicking the corresponding toolbar button.

To create a new transform, make sure that the test setup in which you wish to define the transform is the active setup. This is most easily done by selecting the setup name from the Setup Name combo box at the top of the Transform Editor. In the Transform

edit field, type the name of the desired vector followed by an "equals" sign. Specify the corresponding expression using the dialogue box controls. Alternatively, the expression could be typed by hand using the keyboard instead of the dialogue controls.

HOW TO CREATE A TRANSFORM

1. This example will review the steps necessary to create a transform named Gm, or MOSFET Forward Transconductance. This transform was created in a Gate Threshold test setup in which Id and Vg were measured data vectors.
2. Click the toolbar Transform Editor button to display the Transform Editor dialogue box.
3. In the Setup Name combo box, select the test setup in which you want to define the transform. In this example, the selected test setup was "Gate Threshold".
4. In the Transform edit field, enter the expression: "GM = DELTA(ID)/DELTA(VG)". Expressions can be entered with a "point-and-click" method using the dialogue controls or the expression can be typed directly from the keyboard. The expression can be entered in either upper or lower case. When the SAVE button is selected, the expression will be converted to uppercase characters automatically.
5. Select the SAVE button. This will add the transform definition to the transform list box and the transform vector name (Gm) to the Vectors list box. This allows Gm to be used as a parameter in other transform expressions. Selecting the SAVE button will also clear the Transform edit field.
6. Select the DONE button. This will perform the defined calculations and add the resultant transform vector to the corresponding data window spreadsheet.

To edit an existing transform, double click on the desired transform to display the expression in the Transform edit field. Edit as necessary.



Please note that the Vector name defined in the transform editor cannot end with a number. Win4145 uses numbers appended to a vector name to denote a family of vectors.

HOW TO EDIT A TRANSFORM:

1. Click the toolbar Transform Editor button to display the Transform Editor dialogue box.
2. In the Setup Name combo box, select the test setup in which the transform is defined.
3. Double click on the desired transform expression in the transform list box.
4. Edit the transform expression as necessary using the dialogue box controls or direct from the keyboard.
5. After the expression is edited, click the SAVE button to return the transform expression to the transform list. If the transform vector name is changed, the expression will be added to the list instead of overwriting the original expression.
6. Click the DONE button to close the Transform Editor dialogue box and update the contents of the data window spreadsheet.

EDITING CONSTANTS

The Transform Editor dialogue box includes a Constants list box. The Constants list box displays a list of pre-defined constants. A constant can be inserted into the active transform expression by double-clicking on the constant name.

The Constant Editor provides the capability to edit the name or value of the listed constants as well as add or delete constants from the list. The Constant Editor is displayed by selecting the EDIT CONSTANTS button at the bottom right corner of the Transform Editor dialogue box.

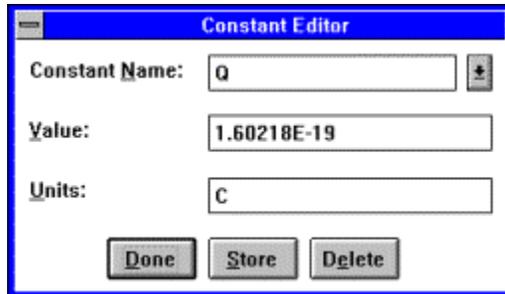


Figure 5-7: The Constant Editor Dialogue Box Provides the Capability to Edit, Add, or Delete Constants from the Constants List Box.

HOW TO ADD, EDIT, OR DELETE A CONSTANT:

1. Open the Transform Editor dialogue box by selecting the appropriate toolbar button.
2. Click the EDIT CONSTANTS button in the bottom-right of the Transform Editor dialogue box. This will display the Constant Editor dialogue box.
3. Enter the name of the constant you wish to add to the list in the Constant Name field. If you wish to edit the value or delete an existing constant, click the Constant Name scroll arrow to display a list of defined constants. Click on the appropriate name.
4. Advance to the Value field using the keyboard TAB key. Specify a numerical value of the new constant. The numerical value can be entered in a scientific format, but engineering units are not acceptable in this field. For example, 2.19E-19 is a valid entry, but 100m is not.
5. If you selected a constant in Step #3, edit the numerical value of the constant as described in Step #4 or delete the constant by selecting the DELETE button. Select the DONE button to close the Constant Editor dialogue box and restore control to the Transform Editor.
6. Advance to the Units field using the keyboard TAB key. Specify the units of the constant, if desired. The units will appear next to the constant name when displayed in the Constants list box. For example, the constant "PI" is listed as "PI [rad]".
7. Select the STORE button to add the new constant to the Constant list box. Select the DONE button to close the Constant Editor dialogue box and restore control to the Transform Editor.

AUTO ANALYSIS EXAMPLES

MOSFET VTH EXTRACTION ALGORITHM

The transform vectors and expression shown below are an example of a MOSFET Turn-On Voltage (V_{th}) extraction.

TRANSFORM EXPRESSION	FUNCTION
$GM = \Delta(ID) / \Delta(VG)$	Calculates GM
$GMMAX = \text{MAXPOS}(GM)$	Finds the position of the maximum value of GM
$GMTFIT = \text{TANFIT}(VG, ID, GMMAX)$	Calculate a tangent line-fit on the VG-ID curve at the VG value corresponding to the position of the maximum value of GM
$VTH = \text{XINT}(GMTFIT)$	VTH equals the x intercept of GMTFIT

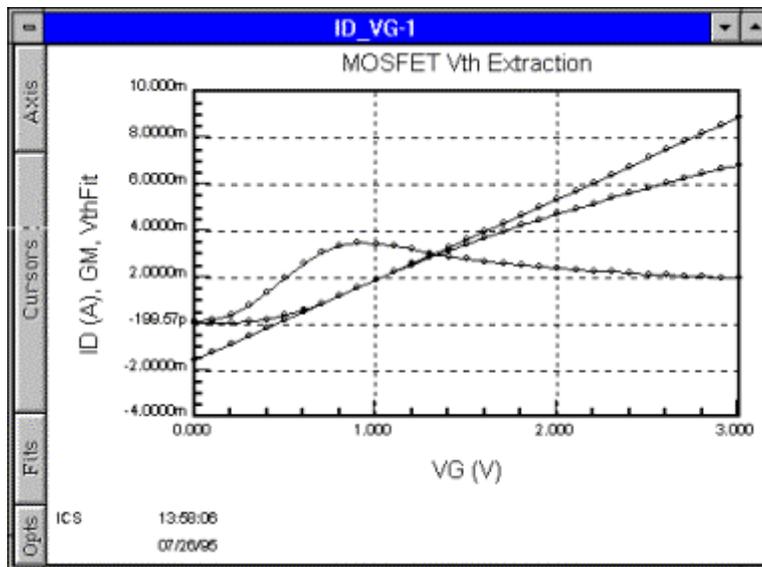


Figure 5-8: Plot of ID, GM, and GMTFIT versus VG.

BIPOLAR VA AND RC EXTRACTION ALGORITHM

The transform vectors and expression shown below are an example of a Bipolar Early Voltage(VA) and Collector output resistance(RC) extraction.

TRANSFORM EXPRESSION	FUNCTION
IDXA=GETPOSD(VC,1.0,1)	Find index where VC = 1.0
ICDATAA=AT(IC,IDXA)	Get IC value when VC = 1.0
VCDATAA=AT(VC,IDXA)	Get VC value when VC = 1.0
IDXB=GETPOSD(VC,3.0,1)	Find index where VC = 3.0
ICDATAB=AT(IC,IDXB)	Get IC value when VC = 3.0
VCDATAB=AT(VC,IDXB)	Get VC value when VC = 3.0
LINE=NORMFIT(VCDATAA,ICDATAA,VCDATAB, ICDATAB, VC)	Calculate normal line fit from point A to point B.
VA=XINT(LINE)	VA equals the x intercept of line
RC=1/SLOPE(LINE)	RC equals 1 over the slope of line

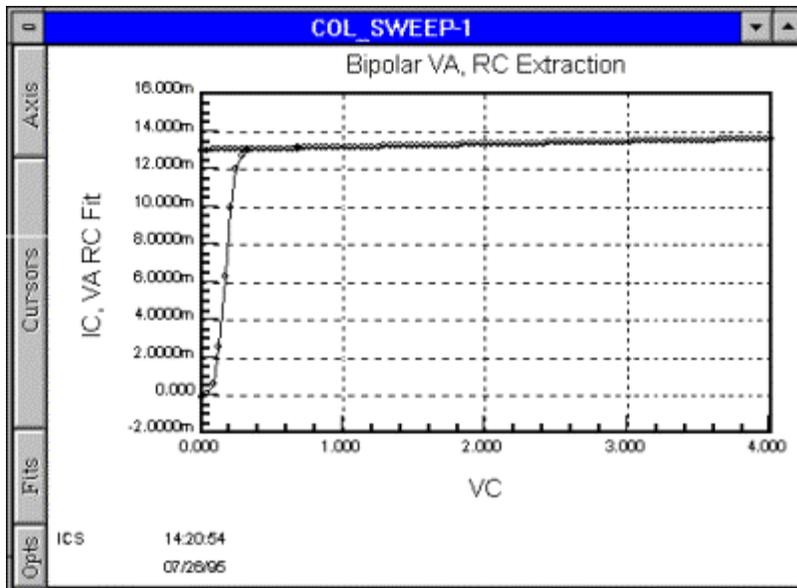


Figure 5-9: Plot of IC and VA RC Fit versus VC.

MOSFET SUB-THRESHOLD SLOPE EXTRACTION ALGORITHM

The transform vectors and expression shown below are an example of the evaluation of the subthreshold slope(S) of a MOSFET.

TRANSFORM EXPRESSION	FUNCTION
IDXA=GETPOSD(ID,1E-7,1)	Find index where ID = 1e-7
IDXB=GETPOSD(ID,1E-5,1)	Find index where ID = 1e-5
LINE=YLOGLINFIT(VG,ID,IDXA,IDXB)	Calculate Least Squares Fit between point A and point B
S=SLOPE(LINE)	S equals the slope of line

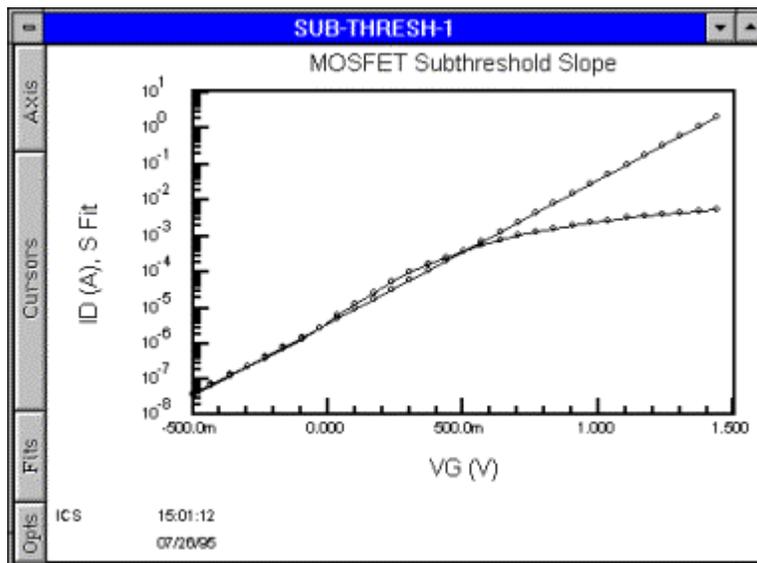


Figure 5-10: Plot of ID and S Fit versus VG.

BIPOLAR hFE DECAY EXTRACTION ALGORITHM

The transform vectors and expression shown below are an example of the evaluation of the hFE decay constant of a bipolar transistor.

TRANSFORM EXPRESSION	FUNCTION
$HFE = IC / IB$	Calculate hFE
$IDXA = GETPOSD(IC, 1E-8, 1)$	Find index where $IC = 1e-8$
$IDXB = GETPOSD(IC, 1E-2, 1)$	Find index where $IC = 1e-2$
$LINE = XYLOGLIN(IC, HFE, IDXA, IDXB)$	Calculate Least Squares Fit between point A and point B
$DECAY = SLOPE(LINE)$	Decay equals the slope of line

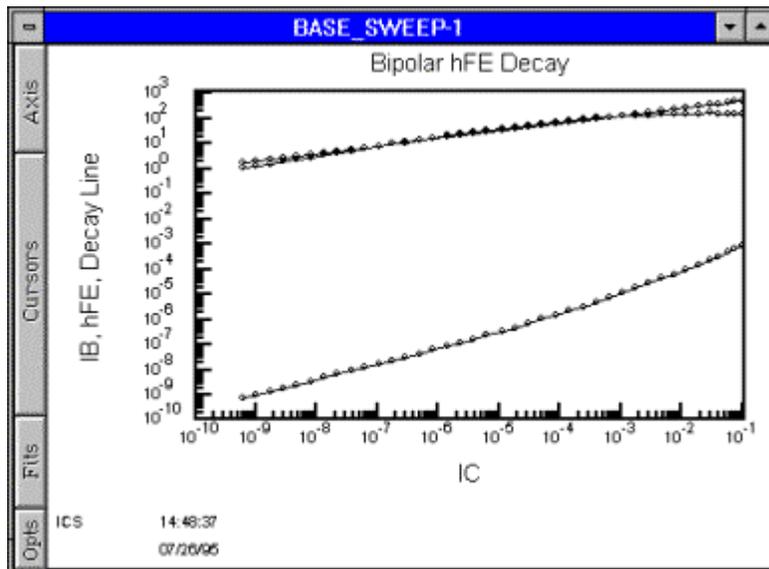


Figure 5-11: Plot of IB , hFE , and Decay Fit versus IC .

TRANSFORM EDITOR FUNCTIONS BY CATEGORY

MATH FUNCTIONS

These functions allow standard math functions to be applied to measured and calculated data.

- ABS
- ARCCOS
- ARCSIN
- ARCTAN
- COS
- EXP
- FRACT
- HOLE
- LN
- LOG
- POW
- SIGN
- SIN
- SQR
- SQRT
- TAN

STATISTICS AND CALCULUS FUNCTIONS

These functions allow statistical and calculus functions to be applied to measured and calculated data.

- AVG
- MEAN
- DELTA
- DIFF
- INTEG
- MAVG
- SUMMV

DATA SEARCH AND EXTRACT FUNCTIONS

These functions allow for searching for values and extracting of values from measured and calculated data.

- AT
- BLDVECT
- EXTRACT
- EXTRANGE
- GETPOSD
- GETPOSU
- INTERP
- INVERT
- MAX
- MAXPOS
- MERGE
- MIN
- MINPOS
- PTCNT

SPECIALIZED FUNCTIONS

See the individual function's definition for a description of the specialized functions application.

- CCORR
- COND
- CONDPP
- GCORR
- GENIDX
- NSUBSI
- NSUBSIB
- RSERIES

FIT PARAMETER FUNCTIONS

These functions allow Fit Parameters to be displayed in data windows or used in other transforms.

- COEFFA
- COEFFB
- FITERR
- SLOPE

- XINT
- YINT

FIT FUNCTIONS FOR LINEAR X AND Y DATA

These fit functions are used to fit straight lines through linear X and Y data.

- EXPFIT
- GRADFIT
- LINFIT
- NORMFIT
- TANFIT

FIT FUNCTIONS FOR LOGARITHMIC X AND LINEAR Y DATA

These fit functions are used to fit a straight line through data which appears straight when the X data is viewed on a logarithmic scale.

- XLOGGRADFIT
- XLOGLINFIT
- XLOGNORMFIT
- XLOGTANFIT

FIT FUNCTIONS FOR LOGARITHMIC X AND Y DATA

These fit functions are used to fit a straight line through data which appears straight when the X and Y data are viewed on a logarithmic scale.

- XYLOGGRADFIT
- XYLOGLINFIT
- XYLOGNORMFIT
- XYLOGTANFIT

FIT FUNCTIONS FOR LINEAR X AND LOGARITHMIC Y DATA

These fit functions are used to fit a straight line through data which appears straight when the Y data is viewed on a logarithmic scale.

- YLOGGRADFIT
- YLOGLINFIT
- YLOGNORMFIT
- YLOGTANFIT

TRANSFER FUNCTION TO MOVE DATA VECTORS FROM ONE SETUP TO ANOTHER

This function allows a setup (the destination setup) to "read" a data vector in another setup (the source setup) within the same project file. This function reads the vector in the source setup at the time that the destination setup is executed. Make sure that the source setup is executed prior to the execution of the destination setup.

NOTE: The brackets [] enclose a single parameter's valid types.

The syntax of this operation is:

[VECTOR | REAL] = [SETUPNAME] | [VECTOR | REAL]

An example is:

CLF = QUASI | C

TRANSFORM EDITOR FUNCTION DEFINITIONS

NOTE: The brackets [] enclose a single parameter's valid types.

FUNCTION: ABS

DEFINITION: Calculates the absolute value of the vector, real, or long value passed.

SYNTAX: [VECTOR | REAL] = ABS ([VECTOR | REAL | LONG])

EXAMPLE: ALLPOS=ABS(IB)

FUNCTION: ARCCOS

DEFINITION: Calculates the arc cosine of the vector, real, or long value passed.

SYNTAX: [VECTOR | REAL] = ARCCOS ([VECTOR | REAL | LONG])

EXAMPLE: ACOS = ARCCOS(IB)

FUNCTION: ARCSIN

DEFINITION: Calculates the arc sine of the vector, real, or long value passed.

SYNTAX: [VECTOR | REAL] = ARCSIN ([VECTOR | REAL | LONG])

EXAMPLE: ASIN = ARCSIN(IB)

FUNCTION: ARCTAN

DEFINITION: Calculates the arc tangent of the vector, real, or long value passed.

SYNTAX: [VECTOR | REAL] = ARCTAN ([VECTOR | REAL | LONG])

EXAMPLE: ATAN=ARCTAN(IB)

FUNCTION: AT

DEFINITION: Extracts a point value from a vector given by the index position.

VECT: Source vector from which to extract the point value.

POS: Index position. Indices start at 1.

SYNTAX: [VECTOR|REAL] = AT(VECT[VECTOR],
POS[REAL|LONG])

EXAMPLE: PT=AT(IB,5)

FUNCTION: AVG

DEFINITION: Calculates the average of all vector point values.

SYNTAX: [VECTOR | REAL] = AVG ([VECTOR])

EXAMPLE: TM=AVG(IB)

FUNCTION: BLDVECT

DEFINITION: Builds a single data vector from the two input data sources.

SYNTAX: [VECTOR] = BLDVECT ([VECT | REAL |
LONG], [VECT | REAL | LONG])

EXAMPLE: NEWVECT=BLDVECT(VTHA, VTHB)

FUNCTION: CCORR

DEFINITION: Calculates the corrected capacitance from a C-G mode measurement. See Nicollian and Brews MOS (Metal Oxide Semiconductor) Physics and Technology.

CVECT: C vector.

GVECT: G vector.

FREQ: Measurement frequency.

OPENVAL: Open system capacitance.

$$CCORR = \frac{(G^2 + w^2 * C^2)C}{(a^2 + w^2 * C^2) - OPENVAL}$$

where

$$w = 2 * \pi * FREQ$$

$$a = G - (G^2 + w^2 * C^2) * R_{series}$$

SYNTAX: [VECTOR | CCORRECTED] = CCORR (CVECT [VECTOR], GVECT[VECTOR], FREQ [REAL|LONG], OPENVAL [REAL|LONG])

EXAMPLE: CCORRECTED=CCORR(C,G,100000,1e-12)

FUNCTION: COEFFA

DEFINITION: Returns the A fit coefficient from the fit passed.

SYNTAX: [VECTOR | REAL] = COEFFA ([FIT])

EXAMPLE: ACOEF=COEFFA(TFIT)

FUNCTION: COEFFB

DEFINITION: Returns the B fit coefficient from the fit passed.

SYNTAX: [VECTOR | REAL] = COEFFB ([FIT])

EXAMPLE: BCOEF=COEFFB(TFIT)

FUNCTION: COND

DEFINITION: This function does the following:

If $A < B$, return C as a single value.

If $A \geq B$, return D as a single value.

Note: If A and/or B are VECTORS, then a comparison is made for each value in the vector

For each index value of the vector, If all $A < B$, return C.

For each index value of the vector, If any $A \geq B$, return D.

SYNTAX: [VECTOR | REAL | LONG] = COND (A [VECTOR | REAL | LONG], B [VECTOR | REAL | LONG], C [VECTOR | REAL | LONG], D [VECTOR | REAL | LONG])

EXAMPLE: TEST=COND(IB,IC,IB,IC)

FUNCTION: CONDPP

DEFINITION: This function does the following for each data point:

If $A < B$, return C.

If $A \geq B$, return D.

SYNTAX: [VECTOR | REAL | LONG] = CONDPP (A [VECTOR | REAL | LONG], B [VECTOR | REAL | LONG], C [REAL | LONG], D [REAL | LONG])

EXAMPLE: TEST=CONDPP(IB,IC,1,0)

FUNCTION: COS

DEFINITION: Calculates the cosine of the vector, real, or long value passed.

SYNTAX: [VECTOR | REAL] = COS ([VECTOR | REAL | LONG])

EXAMPLE: TCOS=COS(IB)

FUNCTION: DELTA

DEFINITION: Calculates the derivative of the vector passed.

SYNTAX: [VECTOR] = DELTA ([VECTOR])

EXAMPLE: DERIV=DELTA(IB)

FUNCTION: DIFF

DEFINITION: Calculates the differential coefficient of A with respect to B.

SYNTAX: [VECTOR] = DIFF (A [VECTOR], B [VECTOR])

EXAMPLE: DIFFCOEF=DIFF(ID,VG)

FUNCTION: EXP

DEFINITION: Calculates the exponential of the vector, real, or long value passed.

SYNTAX: [VECTOR | REAL] = EXP ([VECTOR | REAL | LONG])

EXAMPLE: TEXP=EXP(IB)

FUNCTION: EXPFIT

DEFINITION: Calculates an exponential fit. Individual fit parameters may be retrieved through functions COEFFA, COEFFB, and FITERR. The equation of the curve is: $y = \text{coeffa} * \exp(\text{coeffb} * x)$

X: X vector.

Y: Y vector.

START: Point index at which to start the exponential fit. Must be greater than or equal to 1 and less than or equal to

number of points in the vector.

STOP: Point index at which to stop the exponential fit. Must be greater than or equal to 1 and less than or equal to number of points in the vector.

SYNTAX: [VECTOR | FIT] = EXPFIT (X [VECTOR],
Y[VECTOR], START [REAL|LONG], STOP
[REAL|LONG])

EXAMPLE: EFIT=EXPFIT(VC,IB,2,6)

FUNCTION: EXTRACT

DEFINITION: Extracts a point value from a vector given by the index position.

VECT: Source vector from which to extract the point value.

POS: Index position. Indices start at 1.

SYNTAX: [VECTOR|REAL] =
EXTRACT(VECT[VECTOR],
POS[REAL|LONG])

EXAMPLE: PT=EXTRACT(IB,5)

FUNCTION: EXTRANGE

DEFINITION: Extracts a range of values from a vector as defined by index positions.

VECT: Source vector from which to extract the point value.

POSBEG: Beginning index position. Indices

start at 1.

POSEND: Ending index position. Indices start at 1.

SYNTAX: [VECTOR] = EXTRANGE(VECT[VECTOR],
POSBEG[REAL|LONG],POSEND[REAL|LONG])

EXAMPLE: RNG=EXTRANGE(IB,5,21)

FUNCTION: FITERR

DEFINITION: Returns the fit error associated with the fit passed.

SYNTAX: [VECTOR | REAL] = FITERR ([FIT])

EXAMPLE: ERR=FITERR(TFIT)

FUNCTION: FRACT

DEFINITION: Returns the fractional part of the vector, real, or long value passed.

SYNTAX: [VECTOR | REAL] = FRACT ([VECTOR |
REAL | LONG])

EXAMPLE: TFCT=FRACT(1.31)

FUNCTION: GCORR

DEFINITION: Calculates the corrected conductance from a C-G mode measurement. See Nicollian and Brews MOS (Metal Oxide Semiconductor) Physics and Technology.

CVECT: C vector.

GVECT: G vector.

FREQ: Measurement frequency.

$$\text{GCORR} = \frac{G^2 + w^2 * C^2}{a^2 + w^2 * C^2} a$$

where

$$w = 2 * \pi * \text{FREQ}$$

$$a = G - (G^2 + w^2 * C^2) * R_{\text{series}}$$

SYNTAX: [VECTOR | GCORRECTED] = GCORR (CVECT [VECTOR], GVECT[VECTOR], FREQ [REAL|LONG])

EXAMPLE: GCORRECTED=GCORR(C,G,100000)

FUNCTION: GENIDX

DEFINITION: Generates a vector made up of increasing indices with a range defined by index positions.

POSBEG: Beginning index position. Indices start at 1.

POSEND: Ending index position. Indices start at 1.

SYNTAX: [VECTOR] = GENIDX (POSBEG[LONG],
POSEND[LONG])

EXAMPLE: IDXS=GENIDX(1,21)

FUNCTION: GETPOSD

DEFINITION: Searches a vector from the start index towards the end of the vector for the specified value. The index will be interpolated if an exact match is not found.

VECT: Source vector from which to search for value.

VALUE: Value to search for.

POSBEG: Beginning index position. Indices start at 1.

SYNTAX: [REAL] = GETPOSD (VECT[VECTOR],
VALUE[REAL|LONG],
POSBEG[REAL|LONG])

EXAMPLE: IDXVAL=GETPOSD(VD,0.05,1)

FUNCTION: GETPOSU

DEFINITION: Searches a vector from the start index towards the beginning of the vector for the specified value. The index will be interpolated if an exact match is not found.

VECT: Source vector from which to search for value.

VALUE: Value to search for.

POSBEG: Beginning index position. Indices start at 1.

SYNTAX: [REAL] = GETPOSU (VECT[VECTOR],
VALUE[REAL|LONG],
POSBEG[REAL|LONG])

EXAMPLE: IDXVAL=GETPOSU(VD,0.05,21)

FUNCTION: GRADFIT

DEFINITION: Calculates a gradient fit. Individual fit parameters may be retrieved through functions SLOPE, XINT, and YINT.

PX: Point X value.

PY: Point Y value.

M: Gradient or Slope value.

XV: Vector containing X values.

SYNTAX: [VECTOR | FIT] = GRADFIT (PX [REAL |
LONG], PY[REAL | LONG], M [REAL |
LONG], XV [VECTOR])

EXAMPLE: GFIT=GRADFIT(1.5,2.5,1.25,VC)

FUNCTION: HOLE

DEFINITION: Returns the hole part of the vector, real, or long value passed.

SYNTAX: [VECTOR | REAL] = HOLE ([VECTOR |

REAL | LONG])

EXAMPLE: THLE=HOLE(1.31)

FUNCTION: INTEG

DEFINITION: Calculates the numerical integration of A with respect to B.

SYNTAX: [VECTOR] = INTEG (A [VECTOR],
B[VECTOR])

EXAMPLE: TINTEG=INTEG(IC,VD)

FUNCTION: INTERP

DEFINITION: Calculates the interpolated point for the given vectors and value.

SYNTAX: [REAL] = INTERP (A [VECTOR],
B[VECTOR], AVALUE[REAL|LONG])

EXAMPLE: ICPT=INTERP(VD,IC,0.1)

FUNCTION: INVERT

DEFINITION: Inverts the positions of the vectors values in the vector. i.e. the last value becomes the first and the first becomes the last.

SYNTAX: [VECTOR] = INVERT (A [VECTOR])

EXAMPLE: INVVG=INVERT(VG)

FUNCTION: LINFIT

DEFINITION: Calculates a linear fit. Individual fit parameters may be retrieved through functions SLOPE, XINT, and YINT.

X: X vector.

Y: Y vector.

START: Point index at which to start the linear fit. Must be greater than or equal to 1 and less than or equal to number of points in the vector.

STOP: Point index at which to stop the linear fit. Must be greater than or equal to 1 and less than or equal to number of points in the vector.

SYNTAX: [VECTOR | FIT] = LINFIT (X [VECTOR], Y[VECTOR], START [REAL|LONG], STOP [REAL|LONG])

EXAMPLE: LFIT=LINFIT(VC,IB,2,6)

FUNCTION: LN

DEFINITION: Calculates the natural log of the vector, real, or long value passed.

SYNTAX: [VECTOR | REAL] = LN ([VECTOR | REAL | LONG])

EXAMPLE: TLN=LN(IB)

FUNCTION: LOG

DEFINITION: Calculates the log of the vector, real, or long value passed.

SYNTAX: [VECTOR | REAL] = LOG ([VECTOR | REAL | LONG])

EXAMPLE: TLOG=LOG(IB)

FUNCTION: MAVG

DEFINITION: Calculates the moving average value of A using B number of points for the average.

SYNTAX: [VECTOR] = MAVG(A [VECTOR], B [LONG])

EXAMPLE: BIGVAL=MAVG(IB)

FUNCTION: MAX

DEFINITION: Returns the maximum value of all points in the vector.

SYNTAX: [VECTOR | REAL] = MAX([VECTOR])

EXAMPLE: BIGVAL=MAX(IB)

FUNCTION: MAXPOS

DEFINITION: Returns the index of the vector point which corresponds to the maximum value of all points in the vector. Indices start at 1.

SYNTAX: [VECTOR | LONG] = MAXPOS ([VECTOR])

EXAMPLE: BIGPOS=MAXPOS(IB)

FUNCTION: MEAN

DEFINITION: Calculates the mean of all vector point values.

SYNTAX: [VECTOR | REAL] = MEAN ([VECTOR])

EXAMPLE: TM=MEAN(IB)

FUNCTION: MERGE

DEFINITION: Merges the data in a given X Y data set to a new X or Y vector of the same type.

SYNTAX: [VECTOR] = MERGE (NEWX[VECTOR],
OLDX[VECTOR], OLDY[VECTOR])

EXAMPLE: NEWY=MERGE(NEWVC, VC, IC)

FUNCTION: MIN

DEFINITION: Returns the minimum value of all points in the vector.

SYNTAX: [VECTOR | REAL] = MIN([VECTOR])

EXAMPLE: LOWVAL=MIN(IB)

FUNCTION: MINPOS

DEFINITION: Returns the index of the vector point which corresponds to the minimum value of all points in the vector. Indices start at 1.

SYNTAX: [VECTOR | LONG] = MINPOS ([VECTOR])

EXAMPLE: SMALLPOS=MINPOS(IB)

FUNCTION: NORMFIT

DEFINITION: Calculates a gradient fit. Individual fit parameters may be retrieved through functions SLOPE, XINT, and YINT.

PX1: Point 1 X value.

PY1: Point 1 Y value.

PX2: Point 2 X value.

PY2: Point 2 Y value.

XV: Vector containing X values.

SYNTAX: [VECTOR | FIT] = NORMFIT (PX1 [REAL | LONG], PY1[REAL | LONG], PX2 [REAL | LONG], PY2 [REAL | LONG], XV [VECTOR])

EXAMPLE: NFIT=NORMFIT(1.5,2.5,3.5,4.5,VC)

FUNCTION: NSUBSI

DEFINITION: Calculates an approximation of Nsub for silicon using the successive approximations method using the following input parameters and equations. (See HP Application Note 322 (Analysis of Semiconductor Capacitance Characteristics). This method assumes that $1/C_{ox}$ is negligible with respect to $1/C_{min}$.

Wmax_nm: Max depletion layer width(nm).

NI: Intrinsic carrier concentration/cm³.

T: Temperature (K).

$W_{max-nm} = E_{si} * E_0 * A / C_{min}$

$Fermi = k * T / q * \ln(N_{sub} / n_i)$

$N_{sub} = 4 * Fermi * E_0 * E_{si} / [q * (W_{max_nm})^2]$

(Initial Nsub equals 1e15, maximum of 15 successions)

SYNTAX: [REAL] = NSUBSI (WMAX_NM [REAL | LONG], NI[REAL | LONG], T [REAL | LONG])

EXAMPLE: NSUB=NSUBSI(CSMIN,AREA,NI,TEMP)

FUNCTION: NSUBSIB

DEFINITION: Calculates of Nsub for silicon using the successive approximations method using the following input parameters and equations. (See MOS (Metal Oxide Semiconductor) Physics and Technology

by Nicollan and Brews). This method does not assume that $1/C_{ox} \ll 1/C_{min}$.

CSMIN: Min depletion layer C (F).

CSMAX: Max depletion layer C (F).

AREA: Gate area (cm²).

T: Temperature (K).

$W_{max} = E_{si} * E_o * area * (1/c_{min} - 1/c_{max})$

$Fermi = (k*T/q) * \ln(N_{sub}/n_i)$

$Inversion = 2.0 * Fermi$

$N_{sub} = 2 * E_o * E_{si} * Inversion / (q * W_{max}^2)$

(Initial Nsub equals 1e15, maximum of 15 successions)

SYNTAX: [REAL] = NSUBSI (CSMIN [REAL | LONG],
CSMAX [REAL | LONG], AREA [REAL |
LONG], T [REAL | LONG])

EXAMPLE: NSUB=NSUBSI(CSMIN,CSMAX,AREA,TEMP)

FUNCTION: POW

DEFINITION: Calculates X to the power of Y.

SYNTAX: [VECTOR | REAL] = POW (X [VECTOR |
REAL | LONG], Y [REAL | LONG])

EXAMPLE: IBSQR=POW(IB,2)

FUNCTION: PTCNT

DEFINITION: Calculates the number of points in a vector.

SYNTAX: Size[LONG] = PTCNT(X [VECTOR])

EXAMPLE: PTS=PTCNT(IB)

FUNCTION: RSERIES

DEFINITION: Calculates the corrected capacitance from a C-G mode measurement. See Nicollian and Brews MOS (Metal Oxide Semiconductor) Physics and Technology.

CVECT: C vector.

GVECT: G vector.

FREQ: Measurement frequency.

$$RSERIES=G/(G^2+w^2*C^2)$$

where

$$w=2*pi*FREQ$$

SYNTAX: [VECTOR | RSER] = RSERIES (CVECT [VECTOR], GVECT[VECTOR], FREQ [REAL|LONG])

EXAMPLE: RSER=RSERIES(C,G,100000)

FUNCTION: SIGN

DEFINITION: Returns: 1 if the value passed was greater than or equal to 0, and 0 if the value passed was less than 0.

SYNTAX: [VECTOR | LONG] = SIGN ([REAL | LONG]
)

EXAMPLE: ISPOS=SIGN(-1.5)

FUNCTION: SIN

DEFINITION: Calculates the sine of the vector, real, or long value passed.

SYNTAX: [VECTOR | REAL] = SIN ([VECTOR | REAL
| LONG])

EXAMPLE: TSIN=SIN(IB)

FUNCTION: SLOPE

DEFINITION: Returns the slope of the fit passed.

SYNTAX: [VECTOR | REAL] = SLOPE ([FIT])

EXAMPLE: M=SLOPE(LFIT)

FUNCTION: SQR

DEFINITION: Calculates the square of the vector, real, or long value passed.

SYNTAX: [VECTOR | REAL] = SQR ([VECTOR | REAL
| LONG])

EXAMPLE: TSQR=SQR(IB)

FUNCTION: SUMMV

DEFINITION: Calculates the summation of all values in the vector.

SYNTAX: [VECTOR] = SUMMV ([VECTOR])

EXAMPLE: TSUMM=SUMMV(IB)

FUNCTION: SQRT

DEFINITION: Calculates the square root of the vector, real, or long value passed.

SYNTAX: [VECTOR | REAL] = SQRT ([VECTOR |
REAL | LONG])

EXAMPLE: TSQRT=SQRT(IB)

FUNCTION: TAN

DEFINITION: Returns the tangent of the vector, real, or long value passed.

SYNTAX: [VECTOR | REAL] = TAN ([VECTOR | REAL
| LONG])

EXAMPLE: TTAN=TAN(IB)

FUNCTION: TANFIT

DEFINITION: Calculates a tangent fit at the point given by INDEX. Individual fit parameters may be retrieved through functions SLOPE, XINT, and YINT.

X: X vector.

Y: Y vector.

INDEX: Point index at which to calculate the tangent fit. Must be greater than or equal to 1 and less than or equal to number of points in the vector.

SYNTAX: [VECTOR | FIT] = TANFIT (X [VECTOR], Y[VECTOR], INDEX [REAL|LONG])

EXAMPLE: TFIT=TANFIT(VC,IB,4)

FUNCTION: XINT

DEFINITION: Returns the x intercept from the fit passed.

SYNTAX: [VECTOR | REAL] = XINT ([FIT])

EXAMPLE: XINTER = XINT(LFIT)

FUNCTION: XLOGGRADFIT

DEFINITION: Calculates a X log gradient fit. Individual fit parameters may be retrieved through functions COEFFA, and COEFFB. The equation of the curve is: $y = \text{coeffb} * \log ($

$x) + \text{coeffa}$

PX: Point X value.

PY: Point Y value.

M: Gradient or Slope value.

XV: Vector containing X values.

SYNTAX: [VECTOR | FIT] = XLOGGRADFIT (PX [REAL | LONG], PY[REAL | LONG], M [REAL | LONG], XV [VECTOR])

EXAMPLE: XLGFIT=XLOGGRADFIT(1.5,2.5,1.25,VC)

FUNCTION: XLOGLINFIT

DEFINITION: Calculates a X log linear fit. Individual fit parameters may be retrieved through functions COEFFA, and COEFFB. The equation of the curve is: $y = \text{coeffb} * \log(x) + \text{coeffa}$.

X: X vector.

Y: Y vector.

START: Point index at which to start the linear fit. Must be greater than or equal to 1 and less than or equal to number of points in the vector.

STOP Point index at which to stop the linear fit. Must be greater than or equal to 1 and less than or equal to number of points in the vector.

SYNTAX: [VECTOR | FIT] = XLOGLINFIT (X [VECTOR], Y[VECTOR], START

[REAL|LONG], STOP [REAL|LONG])

EXAMPLE: XLLFIT=XLOGLINFIT(VC,IB,2,6)

FUNCTION: XLOGNORMFIT

DEFINITION: Calculates a X log gradient fit. Individual fit parameters may be retrieved through functions COEFFA, and COEFFB. The equation of the curve is: $y = \text{coeffb} * \log(x) + \text{coeffa}$.

PX1: Point 1 X value.

PY1: Point 1 Y value.

PX2: Point 2 X value.

PY2: Point 2 Y value.

XV: Vector containing X values.

SYNTAX: [VECTOR | FIT] = XLOGNORMFIT (PX1 [REAL | LONG], PY1[REAL | LONG], PX2 [REAL | LONG], PY2 [REAL | LONG], XV [VECTOR])

EXAMPLE: XLNFIT=XLOGNORMFIT(1.5,2.5,3.5, 4.5, VC)

FUNCTION: XLOGTANFIT

DEFINITION: Calculates a X log tangent fit at the point given by INDEX. Individual fit parameters may be retrieved through functions COEFFA, and COEFFB. The equation of the

curve is: $y = \text{coeffb} * \log(x) + \text{coeffa}$

X: X vector.

Y: Y vector.

INDEX: Point index at which to calculate the tangent fit. Must be greater than or equal to 1 and less than or equal to number of points in the vector.

SYNTAX: [VECTOR | FIT] = XLOGTANFIT (X
[VECTOR], Y[VECTOR], INDEX
[REAL|LONG])

EXAMPLE: XLTFIT=XLOGTANFIT(VC,IB,4)

FUNCTION: XYLOGGRADFIT

DEFINITION: Calculates a X log, Y log gradient fit. Individual fit parameters may be retrieved through functions COEFFA, and COEFFB. The equation of the curve is: $y = \text{coeffa} * \exp [\text{coeffb} * \log (x)]$

PX: Point X value.

PY: Point Y value.

M: Gradient or Slope value.

XV: Vector containing X values.

SYNTAX: [VECTOR | FIT] = XYLOGGRADFIT (PX
[REAL | LONG], PY[REAL | LONG], M
[REAL | LONG], XV [VECTOR])

EXAMPLE: XYLGFIT=XYLOGGRADFIT(1.5,2.5,1.25,VC)

FUNCTION: XYLOGLINFIT

DEFINITION: Calculates a X log, Y log linear fit. Individual fit parameters may be retrieved through functions COEFFA, and COEFFB. The equation of the curve is: $y = \text{coeffa} * \exp [\text{coeffb} * \log (x)]$.

X: X vector.

Y: Y vector.

START: Point index at which to start the linear fit. Must be greater than or equal to 1 and less than or equal to number of points in the vector.

STOP Point index at which to stop the linear fit. Must be greater than or equal to 1 and less than or equal to number of points in the vector.

SYNTAX: [VECTOR | FIT] = XYLOGLINFIT (X [VECTOR], Y[VECTOR], START [REAL|LONG], STOP [REAL|LONG])

EXAMPLE: XYLLFIT=XYLOGLINFIT(VC,IB,2,6)

FUNCTION: XYLOGNORMFIT

DEFINITION: Calculates a X log, Y log gradient fit. Individual fit parameters may be retrieved through functions COEFFA, and COEFFB. The equation of the curve is: $y = \text{coeffa} * \exp [\text{coeffb} * \log (x)]$.

PX1: Point 1 X value.

PY1: Point 1 Y value.

PX2: Point 2 X value.

PY2: Point 2 Y value.

XV: Vector containing X values.

SYNTAX: [VECTOR | FIT] = XYLOGNORMFIT (PX1
[REAL | LONG], PY1[REAL | LONG], PX2
[REAL | LONG], PY2 [REAL | LONG], XV
[VECTOR])

EXAMPLE: XYLNFIT=XYLOGNORMFIT(1.5,2.5,3.5,4.5,
VC)

FUNCTION: XYLOGTANFIT

DEFINITION: Calculates a X log tangent fit at the point given by INDEX. Individual fit parameters may be retrieved through functions COEFFA, and COEFFB. The equation of the curve is: $y = \text{coeffa} * \exp [\text{coeffb} * \log (x)]$

X: X vector.

Y: Y vector.

INDEX: Point index at which to calculate the tangent fit. Must be greater than or equal to 1 and less than or equal to number of points in the vector.

SYNTAX: [VECTOR | FIT] = XYLOGTANFIT (X
[VECTOR], Y[VECTOR], INDEX
[REAL|LONG])

EXAMPLE: XYLTFIT=XYLOGTANFIT(VC,IB,4)

FUNCTION: YINT

DEFINITION: Returns the y intercept from the fit passed.

SYNTAX: [VECTOR | REAL] = YINT ([FIT])

EXAMPLE: YINTER=YINT(LFIT)

FUNCTION: YLOGGRADFIT

DEFINITION: Calculates a Y log gradient fit. Individual fit parameters may be retrieved through functions COEFFA, and COEFFB. The equation of the curve is: $y = \text{coeffa} * \exp(\text{coeffb} * x)$

PX: Point X value.

PY: Point Y value.

M: Gradient or Slope value.

XV: Vector containing X values.

SYNTAX: [VECTOR | FIT] = YLOGGRADFIT (PX [REAL | LONG], PY[REAL | LONG], M [REAL | LONG], XV [VECTOR])

EXAMPLE: YLGFIT=YLOGGRADFIT(1.5,2.5,1.25,VC)

FUNCTION: YLOGLINFIT

DEFINITION: Calculates a Y log linear fit. Individual fit parameters may be retrieved through functions COEFFA, and COEFFB. The equation of the curve is: $y = \text{coeffa} * \exp(\text{coeffb} * x)$.

X: X vector.

Y: Y vector.

START: Point index at which to start the linear fit. Must be greater than or equal to 1 and less than or equal to number of points in the vector.

STOP: Point index at which to stop the linear fit. Must be greater than or equal to 1 and less than or equal to number of points in the vector.

SYNTAX: [VECTOR | FIT] = YLOGLINFIT (X
[VECTOR], Y[VECTOR], START
[REAL|LONG], STOP [REAL|LONG])

EXAMPLE: YLLFIT=YLOGLINFIT(VC,IB,2,6)

FUNCTION: YLOGNORMFIT

DEFINITION: Calculates a Y log gradient fit. Individual fit parameters may be retrieved through functions COEFFA, and COEFFB. The equation of the curve is: $y = \text{coeffa} * \exp(\text{coeffb} * x)$.

PX1: Point 1 X value.

PY1: Point 1 Y value.

PX2: Point 2 X value.

PY2: Point 2 Y value.

XV: Vector containing X values.

SYNTAX: [VECTOR | FIT] = YLOGNORMFIT (PX1
[REAL | LONG], PY1[REAL | LONG], PX2
[REAL | LONG], PY2 [REAL | LONG], XV

[VECTOR])

EXAMPLE: YLNFIT=YLOGNORMFIT(1.5,2.5,3.5,4.5,
VC)

FUNCTION: YLOGTANFIT

DEFINITION: Calculates a Y log tangent fit at the point given by INDEX. Individual fit parameters may be retrieved through functions COEFFA, and COEFFB. The equation of the curve is: $y = \text{coeffa} * \exp(\text{coeffb} * x)$

X: X vector.

Y: Y vector.

INDEX: Point index at which to calculate the tangent fit. Must be greater than or equal to 1 and less than or equal to number of points in the vector.

SYNTAX: [VECTOR | FIT] = YLOGTANFIT (X
[VECTOR], Y[VECTOR], INDEX
[REAL|LONG])

EXAMPLE: YLTFIT=YLOGTANFIT(VC,IB,4)

CHAPTER 6: PLOT WINDOWS

WHAT IS A PLOT WINDOW?

Plot windows are graphic presentations of the measured and calculated data in a corresponding data window spreadsheet. Each plot window is dynamically linked to the data window spreadsheet from which it was created. If a data window spreadsheet is automatically updated as a result of re-executing the associated test setup, the plot window will also be automatically updated.



Up to ten plot windows can be defined from a single data window spreadsheet.

Plot windows must be created by the user from the information in a data window spreadsheet. Plot windows are always created as open windows. *Closing* a plot window will delete the window from the desktop. *Minimizing* a plot window will reduce the window to a blue icon positioned along the base of the desktop.

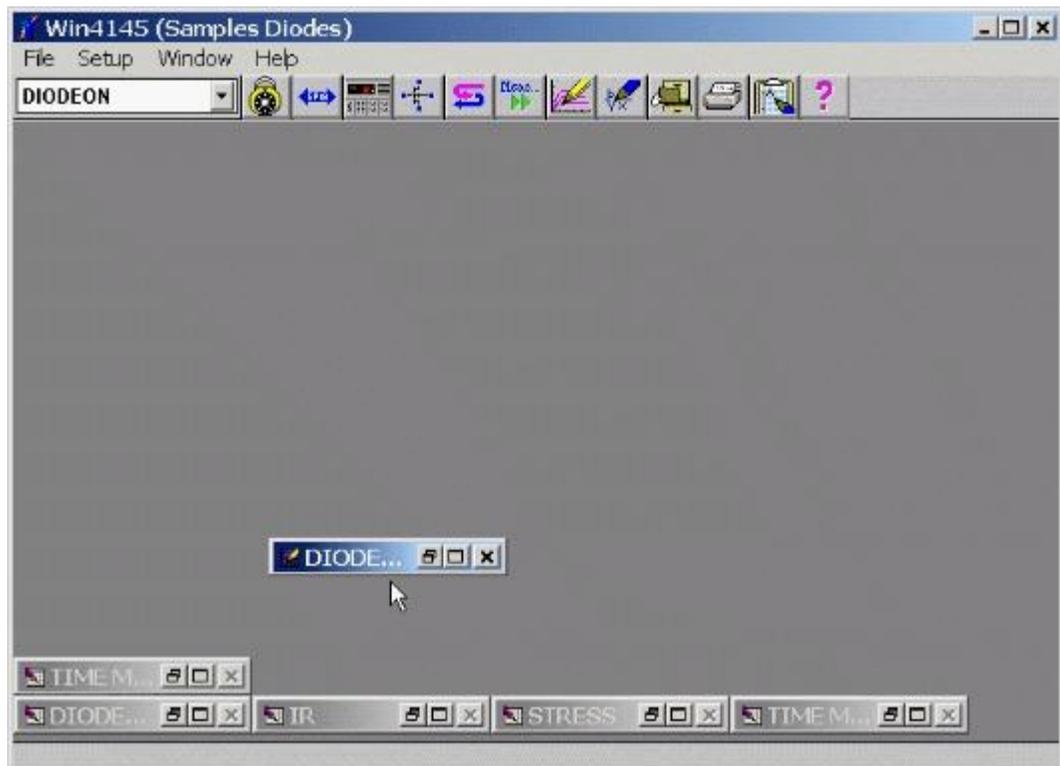


Figure 6-1: The blue (dark) icons shown in this view are minimized plot windows.



A minimized plot window is restored to an open view by clicking twice on the window icon. Clicking once on a plot window icon will open the window control menu. Selecting CLOSE will delete the window from the desktop. The window control menu can also be accessed from an open view. Click once on the window's system menu icon located to the far left of the title bar.



Closed plot windows are permanently destroyed and deleted from the project file.

THE PLOT MENU BAR

When a plot window is active, the menu bar includes pulldown lists particular to plot window functions. To display the plot menu bar, open or select a plot window as the active desktop window, or click once on a plot window icon.

FILE

The FILE menu is available in all three menu bars. It includes all of the file and print control commands that are relevant throughout the entire application.

SETUP

The SETUP menu provides access to the Setup Plot View, Plot Color Setup, and Plot Title dialogue boxes. Other selectable items under the Setup menu include Markers and Lines. All of the Setup menu items are used to edit and format plot windows.

PLOT VIEW

The Setup Plot View dialogue box is displayed automatically when a new plot window is created. The Setup Plot View dialogue box is also accessible from the SETUP menu. This allows the user to edit a plot window definition after the plot window is initially configured. The Setup Plot View dialogue box is displayed by selecting SETUP/PLOT VIEW from the plot mode menu bar.

COLORS

The Plot Color Setup dialogue box is accessible from the SETUP menu as well as the plot window toolbar. This dialogue allows the user to edit the colors of the plot items. The Plot Color Setup dialogue box is displayed by selecting SETUP/COLORS from the plot mode menu bar.

TITLES

The Plot Titles dialogue box is accessible from the SETUP menu as well as the plot window toolbar. This dialogue allows the user to edit the Title, X axis label, and Y axis labels. The Plot Titles dialogue box is displayed by selecting SETUP/TITLES from the plot mode menu bar.

MARKERS

The MARKERS menu item allows the user to turn on and off the display of the circular markers which designate points on the plot curves. The MARKERS function is turned on and off by selecting SETUP/MARKERS from the plot mode menu bar. A check mark designates the function is on.

LINES

The LINES menu item allows the user to turn on and off the display of the line which connects the points on the plot curve. The LINES function is turned on and off by selecting SETUP/LINES from the plot mode menu bar. A check mark designates the function is on.

WINDOW

The WINDOW dropdown list is available in all three menu bars. This is a Windows feature that includes the standard window and desktop arrangement commands. The WINDOW menu includes the NEW PLOT, UNHIDE DATA WINDOW, and COPY PLOT functions that are specific to Win4145. The NEW PLOT and COPY PLOT functions can also be accessed from the toolbar.

HELP

The HELP dropdown list is also available in all three menu bars. The HELP menu selection is used to invoke the On-Line Help Program.

PLOT WINDOW TOOLS

When focus is set to a plot window, the plot mode menu bar is displayed under the Win4145 title strip at the top of your display. In addition to the plot mode menu bar, each plot window includes a tool ribbon along the left edge of the window. The menu bar and tool ribbon includes all of the formatting and analysis tools available in an Win4145 plot window environment. Some of the formatting and analysis tools are accessible from both the menu bar and the plot window tool ribbon. Below is a summary of the plot window tools available with Win4145.

AXIS

Y₁/Y₂ PLOTTING CAPABILITY

1. Autoscale or Manually Designate Axis Ranges
2. LIN/LOG Scaling Independently Controlled for Each Axis
3. Invert ON/OFF Independently Controlled for Each Axis
4. Gridlines ON/OFF Independently Controlled for Each Axis
5. Data Markers: ON/OFF
6. Data Lines: ON/OFF

CURSORS

FIVE CURSORS ARE AVAILABLE PER PLOT WINDOW

1. Five Cursors Independently Configured to Y_1 , Y_2 , or OFF.
2. Cursor Positions Automatically Displayed on Plot Window
3. Cursor Crosshairs: ON/OFF
4. Active Cursor to MAX
5. Active Cursor to MIN
6. Active Cursor Direct to Point.
7. Align Cursor Pair
8. Center Plot at Cursor
9. DataPoint Tracking: ON/OFF
10. Cursor Pair X/Y Deltas
11. Automatically Displayed on Plot Window, When Selected

CURVE FITS

DISPLAY TWO CURVE FITS PER PLOT WINDOW

Five Curve Fit Routines Available:

1. Normal Fit
2. Linear Fit
3. Exponential Fit
4. Tangent Fit
5. Gradient Fit

Curve Fit Parameters Automatically Displayed on Plot Window

FORMAT/DISPLAY OPTIONS

1. Display Legend
2. Display Source Unit Test Conditions
3. Plot Setup
4. Add Titles and Labels
5. Plot Title
6. X-Axis Label
7. Y₁-Axis Label
8. Y₂-Axis Label
9. Comment Line
10. Zoom In View
11. Zoom Out View
12. Fixed Line Fit Display
13. Edit Plot Window Colors
14. Independently Configure each Curve and Plot Object Color
15. Fourteen Colors Available
16. Configure Plot Window in Monochrome Display (for Hard copy Output)
17. Four Memories for Overlay Plots



When a plot window is created, the plot window will be formatted according to the default configuration listed in Table 2.

Feature		Default
Axes	Axis Ranges	Autoscaled
	X-Axis Display	Linear
	Y1 -Axis Display	Linear
	Y2 - Axis Display (if present)	Linear
	Axis Labels	Data Vector Name
	Axis Inversion	OFF
	Gridlines	OFF
	Data Markers	ON
	Data Lines	ON
Cursors	DataPoint Tracking	ON
	Crosshairs	OFF
	Deltas	OFF
Format	Display Legend	OFF
	Display SMU Test Conditions	OFF
	Plot Title	"Title"
	Comment Line	OFF
	Axis Labels	See Above

Table 2 – Default Plot Window Settings

CREATING A PLOT WINDOW

A plot window is created by selecting the NEW PLOT toolbar button or by selecting WINDOW/NEW PLOT from the plot mode menu bar. When the user creates a new plot window, Win4145 will display the Plot Data dialogue box. The Plot Data dialogue box requires the user to specify two categories of information:

1. **Select the data vectors** to include in the plot and identify which data vector is the independent variable (x-axis), and which data vector(s) are the dependent variables (y-axis).
2. **Select between a Y1 and Y2 scale assignment** for each dependent variable included in the plot.

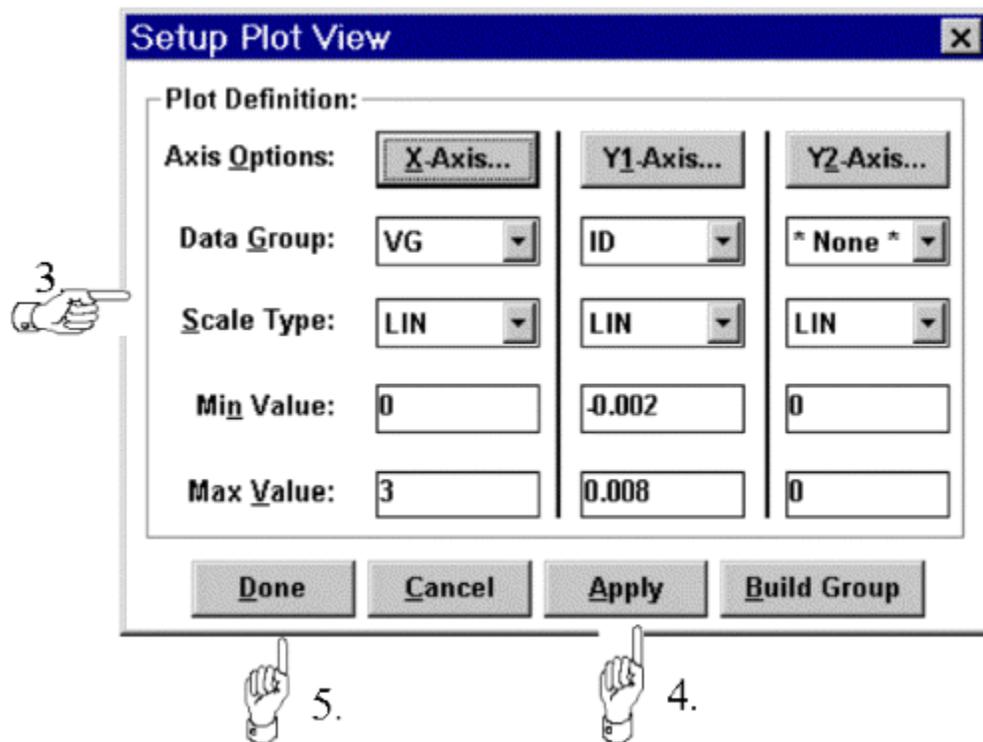


Figure 6-2: How to Create a New Plot

HOW TO CREATE A NEW PLOT:

1. Select the test setup in the toolbar Setup window corresponding to the applicable data window spreadsheet. The associated data window spreadsheet can be displayed in full view, minimized, or hidden.
2. Click the New Plot toolbar button or optionally select WINDOW/NEW PLOT from the plot mode menu bar. This will open an untitled plot window and the Setup Plot View dialogue box.
3. The Setup Plot View dialogue box will include a lists of data vectors measured in the corresponding test setup, in the DATA GROUP controls. Identify the X, Y1, and Y2 variables of the plot by selecting the desired variable from the DATA GROUP controls.
4. Click on the Apply button. This will create the plot window without closing the Plot Data dialogue box. When Win4145 creates the plot window, the axis ranges will be autoscaled and configured according to a linear-linear plot style. These specifications can be changed later from the Plot Axes and Plot Style dialogue boxes.
5. Click on the Close button to close the Plot Data dialogue box.



When a plot window is created, an autoscaling routine determines the lower and upper limits of each axis. Subsequent plot window updates WILL NOT be autoscaled. When the re-execution of the corresponding test setup returns data beyond the present range of one or more axes, the user has the option of autoscaling the updated plot window by selecting the appropriate button included in the plot window tool ribbon.

EDITING A PLOT WINDOW DEFINITION

A plot window definition can be edited after the plot window is created. This is done by re-opening the Setup Plot View dialogue box and re-specifying the data vector positions and/or axis assignments.



The Setup Plot View dialogue box is displayed by selecting SETUP/PLOT VIEW from the plot mode menu bar.

HOW TO EDIT A PLOT WINDOW DEFINITION:

1. Make sure the plot in question is the active window. If it isn't, click once anywhere in the plot window.
2. Open the Setup Plot View dialogue box by selecting SETUP/PLOT VIEW from the plot mode menu bar.
3. Re-specify the X, Y1, and Y2 variables of the plot by selecting the desired variable from the DATA GROUP controls.



To change the selection of data vectors included in the plot simply re-specify the X, Y1, and Y2 variables of the plot in the DATA GROUP controls.

BUILDING DATA GROUPS FOR PLOT DISPLAYING

A data group is a group of data vectors to be plotted on a single Y axis. Normally only one data vector is plotted for every Y axis but if a plot requires more than one data variable to be plotted on the Y1 or Y2 axis then the use of a data group is required.



Data Groups allow users to plot more than one data vector against the Y1 and Y2 axis.

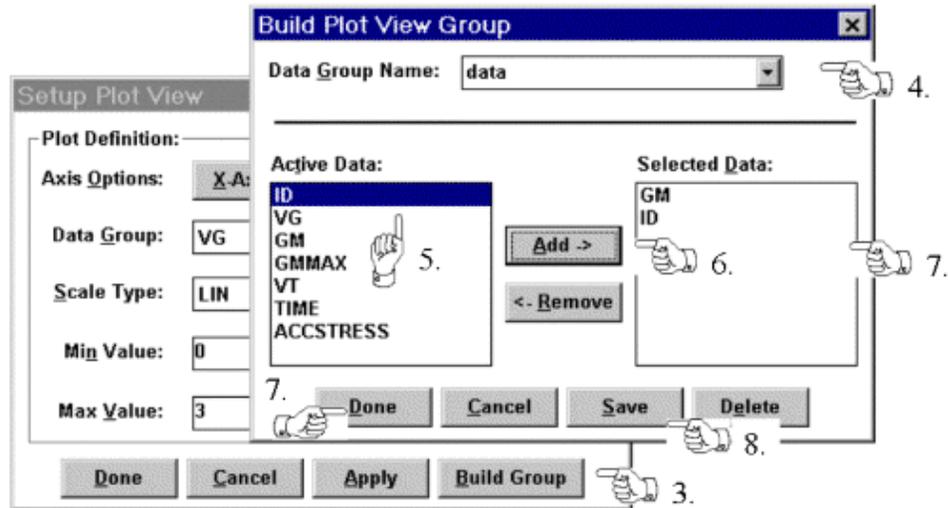


Figure 6-3: How to Create a Plot Data Group.

HOW TO CREATE A PLOT DATA GROUP:

1. Make sure the plot in question is the active window. If it isn't, click once anywhere in the plot window.
2. Open the Setup Plot View dialogue box by selecting SETUP/PLOT VIEW from the plot mode menu bar.
3. Click on the BUILD GROUP button to display the Build Plot View Group dialogue box.
4. Specify the name for the new Plot Data Group.
5. Highlight the data vector name to be added to the group in the ACTIVE DATA list.
6. Click on the ADD button to add the data vector to the group.
7. The added data vector's name will appear in the SELECTED DATA list which is the list of data vectors contained in the Plot Data Group.
8. Click on SAVE to save the data group and to continue editing the group or click on DONE to save the data group and to return to the Setup Plot View dialogue box.

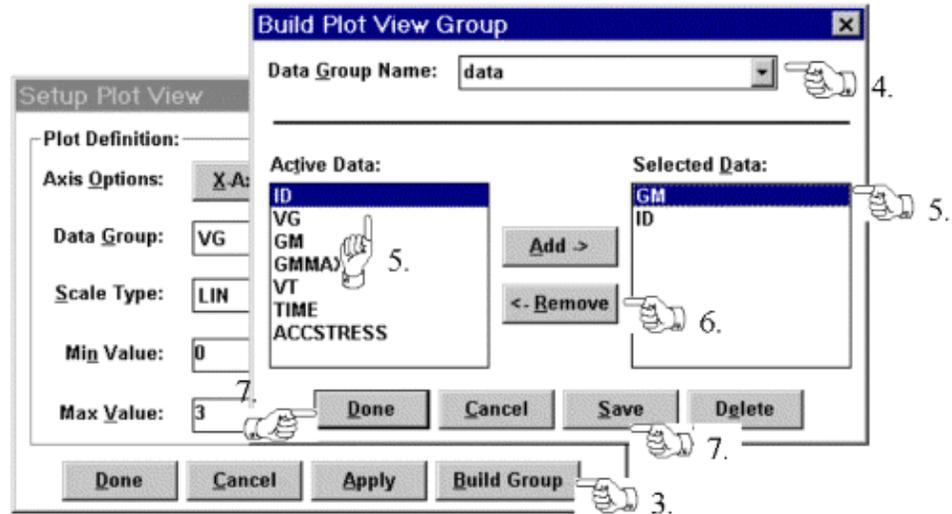


Figure 6-4: How to Edit or Delete a Plot Data Group.

HOW TO EDIT OR DELETE A PLOT DATA GROUP:

1. Make sure the plot in question is the active window. If it isn't, click once anywhere in the plot window.
2. Open the Setup Plot View dialogue box by selecting SETUP/PLOT VIEW from the plot mode menu bar.
3. Click on the BUILD GROUP button to display the Build Plot View Group dialogue box.
4. Specify the name of the Plot Data Group to edit or delete in the DATA GROUP NAME control.
5. Highlight the data vector name to be deleted from the group in the SELECTED DATA list.
6. Click on the REMOVE button to remove the data vector from the group. To delete the data group repeat steps 5 and 6 until all data vectors are removed from the SELECTED DATA list.
7. Click on SAVE to save the data group and to continue editing the group or click on DONE to save the data group and to return to the Setup Plot View dialogue box.

THE PLOT WINDOW TOOL RIBBON

Each plot window includes a vertical tool ribbon along the left side of the window. The tool ribbon provides easy access to much of the formatting and analysis capability relevant to a plot window environment.

The plot window tool ribbon consists of four buttons: Axis, Cursors, Fits, and Options. Each button displays or hides a secondary toolset. Each of the four secondary toolsets group the controls related to the tool ribbon category.

When a secondary toolset is displayed by selecting the corresponding tool ribbon button, the toolset will be painted on top of the plot window. Displaying a secondary toolset will not resize the plot area. Displaying a secondary toolset may temporarily block the view of a relevant plot area. To restore the view, hide the opened toolset(s) when their functionality is no longer needed. This is done by clicking the corresponding tool ribbon button that was used to display the toolset.



The plot window tool ribbon is designed to occupy the entire vertical border of a default-sized plot window. The tool ribbon is a static size. It will not be rescaled as a plot window is altered in size or proportion.

THE AXES TOOLSET

The Axes toolset includes the controls necessary to autoscale the plot display as well as designate between logarithmic or linear plot axes scaling styles.

AUTOSCALE

When a plot window is created, Win4145 will autoscale each axis according to the available data. If the range of the available data changes during subsequent executions of the test setup, the axes ranges will not be changed automatically but will retain their original limits. When the user selects the plot window AUTOSCALE

button, Win4145 will recalculate the axis ranges according to the corresponding data.

LIN/LOG AXES STYLES

Below the AUTOSCALE button are three toggle-buttons used to designate the scaling style of the respective axis: X, Y₁, and Y₂. As a default, all three axes are configured to a linear style. Any of the three axis buttons can be toggled to the alternative style by clicking once on the button. Each axis button displays an icon depicting the active style selection. When the button is toggled, the icon changes to the new style.

THE CURSORS TOOLSET

Up to five cursors are available in each plot window. Cursors can be assigned to any combination of Y₁ or Y₂ positions. The cursors and cursor tools are accessed from the Cursors toolset. The Cursors toolset is displayed by clicking the vertical CURSORS button in the plot window tool ribbon. The Cursors toolset consists of three sections: a static display of the cursor shapes, the cursor assignment buttons, and the cursor tools.

The left edge of the Cursors toolset includes a static display of the five cursor shapes. The five cursor shapes are yellow and black in appearance and are displayed in what appear to be "depressed" buttons. These are not really buttons. The cursor shapes are displayed for reference only. Nothing will happen if you click on the cursor shapes.

Next to each of the five cursor shapes is a cursor assignment button. As a default, each button will be set to "OFF". The cursor assignment buttons allow the user to assign each cursor to either the Y₁ or Y₂ axis or turn the cursor OFF. This is done by clicking once on the assignment button to cycle to the next option. By repeatedly clicking on the assignment button, the user can cycle the cursor status through Y₁-Y₂-OFF.

The right edge of the Cursors toolset includes a set of buttons. Each button represents a particular tool that is related to the functionality of the cursors. From top to bottom, the cursor tools are:

1. Cursor Crosshairs: ON/OFF
2. Active Cursor to MAX
3. Active Cursor to MIN
4. Align Cursor Pair
5. Data Point Tracking: ON/OFF
6. Cursor Direct
7. Cursor Pair X/Y Deltas
8. Interpolation ON/OFF
9. Center Plot at Cursor

A review of each tool is presented later in this section.

ASSIGNING CURSORS TO A PLOT WINDOW

Up to five cursors can be assigned to each plot window. Cursors can be assigned to any combination of Y_1 or Y_2 positions. After selecting the desired cursor from the Cursors toolset, the cursor can be positioned by clicking once at the desired plot window location. More information about cursor positioning is presented later in this chapter.



Once assigned to a plot window, cursor assignments will be stored with the plot window until the cursors are turned "OFF" by the user. This is true even if the project file is closed and later re-opened, or Win4145 is shut off and later re-started.



When a cursor is added to the plot window, the graph will be re-sized to accommodate the Cursor and Curve Fit tables displayed along the bottom of the plot window. The Cursor and Curve Fit tables will be displayed as long as one or more cursors are assigned to the plot window.

HOW TO ASSIGN CURSORS TO A PLOT WINDOW

1. Open the Cursors toolset by clicking the tool ribbon CURSORS button.
2. The five cursors will be identified by the five cursor shapes displayed in the Cursors toolset. Any of the five cursors can be assigned in any order. Locate the cursor assignment button next to the desired cursor shape. As a default, all five assignment buttons will display "OFF".
3. To assign the respective cursor to the Y_1 axis, click the assignment button once. To assign the cursor to the Y_2 axis, click the assignment button again. Click the assignment button a final time to turn OFF the cursor. If the plot window does not include a Y_2 axes, the assignment button will cycle between Y_1 and OFF.
4. When a cursor is added to the plot window, the graph will be re-sized to accommodate the Cursors and Curve Fit tables displayed along the bottom of the plot window. The Cursors and Curve Fit tables will be displayed as long as one or more cursors are assigned to the plot window.
5. To move the cursor to a specific plot window position, move the mouse pointer to the center of the cursor. Click and hold the left mouse button while dragging the cursor to the desired location. Do not release the left mouse button until the cursor is correctly positioned. Additional positioning techniques are presented later in this section.
6. After you are done assigning cursors to the plot window, clear the Cursors toolset from the plot window display by clicking the tool ribbon CURSORS button.

THE CURSOR TABLE

When one or more cursors are assigned to the plot window, the plot window will be resized to accommodate the Cursor Table and two Curve Fit tables displayed underneath the plot area.

The Cursor Table displays each of the five cursor shapes and one triangle symbol representing the results of the Delta function. The remaining area of the Cursor table consists of two columns: X and Y. When any combination of cursors is assigned to the plot window, the X-axis and Y-axis positions are reported in the Cursor table next to the corresponding cursor shape. The positions reported in the Cursor table are dynamic. This means that as the cursors are moved within the plot area, the values displayed in the table are updated continuously.



When a plot window is printed in hard copy or copied to the Windows® Clipboard, the Cursor table will be included in the result.

The last row of the Cursor table displays the result of the Delta function. When the Delta function is activated, the X-axis and Y-axis columns display the DX and DY values corresponding to the selected cursor pair. The Delta function will be addressed in more detail later in this chapter.

CONTROLLING CURSOR POSITIONS

Cursors can be moved around the plot window after the cursors are initially positioned. There are three different ways to reposition a cursor. Each of the three methods work whether the tracking option is ON or OFF. (The Tracking function is presented later in this section).

1. Click and hold the cursor shape using the left mouse button. Drag the cursor to the desired location.
2. Click and release the left mouse button. Move the mouse to the desired location. Click and release the left mouse button again. The cursor will be redrawn at the new position.
3. Position the mouse on the cursor shape. Click and release the left mouse button. Use the keyboard arrow keys to move the cursor to the desired location.

The Tracking function forces the cursor to "lock-on" to the nearest data point. When the Tracking function is active, the cursor will remain at the same relative curve position when the plot window is refreshed. This is true even if the data points of the new curve change values. For example, if a cursor is positioned at the 32nd data point on the third of four curves, the cursor will remain positioned at the 32nd data point on the same curve even if the measurement is re-executed and the data point changes value.



When the Tracking function is active, the cursor will remain at the same relative curve position when the plot window is refreshed. This is true even if the data points of the new curve change values.

DELETING CURSORS FROM A PLOT WINDOW

Cursors are deleted from a plot window by clicking on the cursor assignment button in the Cursors toolset until the button displays the word "OFF".

HOW TO DELETE CURSORS FROM A PLOT WINDOW

1. Open the Cursors toolset by clicking the tool ribbon CURSORS button.
2. The five cursors will be identified by the five cursor shapes displayed in the Cursors toolset. Locate the cursor assignment button next to the cursor shape representing the cursor you wish to remove.
3. The assignment button will read either Y_1 or Y_2 , depending upon the axis at which the cursor is positioned. Click the assignment button until the button displays the word "OFF".
4. Each of the five cursors must be deleted individually. After the last cursor is removed from the plot, the cursor and curve fit tables will be removed from view and the plot area will be resized to occupy a larger area of the plot window.

CURSOR FUNCTIONS

The right edge of the Cursors toolset includes a set of buttons. Each button represents a particular tool that is related to the functionality of the cursors. The Cursor Functions are summarized below as they appear from top to bottom in the Cursors toolset:

1. **Cursor Crosshairs ON/OFF:** A pair of perpendicular gridlines can be displayed through each cursor. This function sets the crosshair status of all the cursors. The crosshair option cannot be individually controlled for each cursor.
2. **Active Cursor to MAX:** This function moves a selected cursor to the curve's maximum Y-axis value.
3. **Active Cursor to MIN:** This function moves a selected cursor to the curve's minimum Y-axis value. Each of the five cursors must be deleted individually.
4. **Align Cursor Pair:** The alignment function positions two cursors at the same X-axis location. This function is most useful when it is necessary to align a pair of cursors assigned to different curves.

5. **Data Point Tracking:** The tracking option will assign each newly positioned cursor to the nearest data point.
6. **Cursor Direct:** Prompts the user to enter a curve name and an X value at which to place the active cursor.
7. **Cursor Pair X/Y Deltas:** The Delta function calculates the difference in X-axis and Y-axis values between two selected cursors. The results of this function are reported in the last row of the plot window Cursor table.
8. **Interpolation ON/OFF:** Turns the linear interpolation function on or off.
9. **Center Plot at Cursor:** Causes the plot to be redrawn with the active cursor as the center of the plot.

CURSOR CROSSHAIRS ON/OFF

Crosshairs are perpendicular gridlines that can be displayed through the center of each cursor. Crosshairs run the entire horizontal and vertical length of the plot window. When selected, the Crosshairs option is applied to all of the cursors.

The Crosshairs button is a toggle-control. When the Crosshairs button is depressed, the function is active. Crosshairs will be displayed through any cursors already assigned to the plot window and any future cursor assignments. To turn off the crosshair displays, click the button again.

ACTIVE CURSOR TO MAX

The Maximum Value function moves a selected cursor to the curve's maximum Y-axis value. This function is applied by clicking once on the plot window cursor shape (this establishes the cursor as the active cursor) followed by the MAX button in the Cursors toolset.

ACTIVE CURSOR TO MIN

The Minimum Value function moves a selected cursor to the curve's minimum Y-axis value. This function is applied by clicking once on the plot window cursor shape (this establishes the cursor as the active cursor) followed by the MIN button in the Cursors toolset.

ALIGN CURSOR PAIR

The alignment function positions two cursors at the same X-axis location. This function is most useful when it is necessary to align a pair of cursors assigned to different curves.

Selecting the Align button in the Cursors toolset will display a Cursor Selection strip to the immediate right of the Align button. The Cursor Selection strip is simply a row of five buttons in which each button corresponds to one of the five cursors. Each button displays a black-and-white image of the corresponding cursor. Identify the cursor pair you wish to align by clicking on the appropriate pair of buttons. Make sure that the cursor positioned at the target X-axis location is selected first! When a button is depressed, the cursor shape is displayed in color to identify the cursor as selected. After the second cursor is selected, the plot window cursor pair will be aligned to the X-axis location corresponding to the cursor selected first. Click the toolset Align button to remove the Cursor Selection strip from view.



The cursor designated first identifies the X-axis location to which the second cursor will be repositioned.



The alignment function cannot be used to align cursors with respect to the Y-axis.

TRACKING

The tracking option will assign each newly positioned cursor to the nearest data point. Toggle the Tracking switch to turn the option ON/OFF. The tracking option is global to all the cursors but will not update the location of any cursors positioned before Tracking was selected. After Tracking is selected, click once on each previously positioned cursor to relocate the cursor to the nearest data point.

The tracking option limits *where*, not *how*, a cursor can be positioned. The tracking option assigns a cursor to the datapoint nearest the position indicated by the mouse pointer. When Tracking is not selected, a cursor can be positioned anywhere in the plot window, even if the location is not on a curve. In either case cursors can still be moved around the plot window as described in *Controlling Cursor Positions*.

CURSOR DIRECT

A cursor may be placed in a desired location on a plot window curve by selecting the CURSOR DIRECT button and selecting the name of the curve and the X coordinate of the desired cursor location.

DATA POINT X/Y DELTAS

If two or more cursors are applied to a single plot window, the differences between the X-axis and Y-axis locations can be calculated between a selected cursor pair. The result of this function is displayed in the last row of the Cursor table displayed underneath the plot area.

Selecting the Delta button in the Cursors toolset will display a Cursor Selection strip to the immediate right of the Delta button. The Cursor Selection strip is simply a row of five buttons in which each button corresponds to one of the five cursors. Each button displays a black-and-white image of the corresponding cursor. Identify the cursor pair of interest by clicking on the appropriate pair of buttons. When a button is depressed, the cursor shape is displayed in color to identify the cursor as selected. After the second cursor is selected, the corresponding delta values will be displayed in the plot window Cursor table. Click the toolset Delta button to remove the Cursor Selection strip from view.

To remove the delta calculations from the Cursor table, display the Cursor Selection strip by clicking the toolset Delta button. Click the active cursor selections so that the cursor images are restored to black-and-white. Click the Delta button a second time to remove the Cursor Selection strip from display. The delta calculations are no longer shown.



Delta calculations are determined according to the following precedence: (First Designated Cursor) - (Second Designated Cursor).

LINEAR INTERPOLATION ON/OFF

The Interpolation button turns on and off the linear interpolation of 25 intermediate points between two measured points.

CENTER PLOT AT CURSOR

Selection of the Center Plot at Cursor button causes the current plot to be redrawn with the active cursor at the center of the plot.

THE CURVE FIT TOOLSET

The Curve Fit toolset provides the capability to apply up to two curve fit routines to a plot window. The Curve Fit toolset is displayed by clicking the corresponding tool ribbon button. The choice of curve fitting algorithms includes a cursor-to-cursor normal fit, a linear least squares calculation, an exponential calculation, a tangent line calculation, and a gradient calculation.



Up to two curve fit routines can be applied to a single plot window. Curve fit algorithms include: a cursor-to-cursor normal fit, a linear least squares calculation, an exponential calculation, a tangent line calculation, and a gradient calculation.

ADDING CURVE FITS TO A PLOT WINDOW

The required number of cursors must be assigned to the plot window before configuring the curve fit(s). After the cursors are positioned, up to two curve fits are configurable using the controls in the Curve Fit toolset. The Curve Fit toolset consists of two large buttons: the first button is designated "Curve Fit #1", and the second button is designated "Curve Fit #2". Selecting either button will display another set of controls. These controls are used to designate the relevant cursors along with the curve fit type you wish to apply.

HOW TO APPLY A CURVE FIT ROUTINE TO A PLOT WINDOW:

1. Click the vertical Curve Fit button in the plot window tool ribbon. This will display a pair of buttons. The first button is labeled "Curve Fit #1" and the second button is labeled "Curve Fit #2".
2. Select either of the Curve Fit toolset buttons. This will display a secondary set of curve fit controls used to designate the curve fit cursors, fixed line on/off function, and curve fit type. The first five buttons of the secondary controls correspond to the five available cursors. This section of the controls is called the Cursor Selection strip. The next button turns on or off the fixed line function which causes a line fit to not update with new data. The last five buttons correspond to the five available curve fit types. This section of the controls is called the Curve Fit Selection strip.
3. Designate the cursors through which the curve fit will be applied by depressing the appropriate cursor buttons in the Cursor Selection strip. When a cursor selection button is depressed, the cursor image pictured on the button will change from black-and-white to color to show that the cursor is "selected". Only cursors that are assigned to the plot window can be selected from the Cursor Selection strip. Up to two cursors can be selected at a time. If you select a third cursor, the first designation will be "unselected". Each curve fit type, except for the tangent fit, requires two cursor selections. The tangent and gradient fits only require a single cursor selection.

4. Designate the curve fit type by depressing the appropriate button in the Curve Fit Selection strip. From left to right, the five buttons are: cursor-to-cursor normal fit, linear fit, exponential fit, tangent fit, and gradient fit. After the curve fit type is selected, the curve fit line will appear in the plot window and the curve fit parameters will appear in the respective Curve Fit table underneath the plot area. In the case of the gradient fit the user will be prompted for the desired gradient value.
5. Click on the Fixed fit button if it is required that the fit does not move with updated data.
6. The Cursor and Curve Fit Selection strip can be hidden from view by clicking the corresponding Curve Fit button. The Curve Fit toolset can be hidden from view by clicking the tool ribbon Curve Fit button.
7. To add a second curve fit to the plot window, select the alternative Curve Fit button from the Curve Fit toolset and repeat Steps #3 through #6.
8. To remove a curve fit from the plot window, click the tool ribbon Curve Fit button and select either the Curve Fit #1 button or the Curve Fit #2 button, whichever is appropriate. This will display the Cursor and Curve Fit Selection strip. To turn OFF the fit line, click the curve fit selection button corresponding to the fit type. An alternative fit type can be selected by clicking an alternative fit type button. To turn OFF the fit completely, unselect the corresponding cursors in the selection strip.

NORMAL FIT

The Normal fit calculates a straight line through two cursor locations. The line's equation is calculated using a Chi-Squared Linear Regression model that considers only the two specified cursor points. The two cursors used in this calculation can be located anywhere in the plot window. If a family of curves is displayed in the plot window, the cursors can be located on different curves.

The slope, X_{int} , and Y_{int} are reported in the respective Curve Fit Table. For a description of the chi-square function, see *Linear Curve Fit*.

LINEAR FIT

The Linear fit calculates a straight line through a range of data bordered by two cursors. The line's equation is calculated using a Chi-Squared Linear Regression model that considers the two cursor points as well as the data points between the specified cursor locations. This calculation will only provide a meaningful result if the data points considered in the model are from the same measurement sweep. In other words, in a plot window displaying a family of curves, the specified cursors should be on the same curve.

The Chi-Squared Linear Regression model plots a straight line, $y(x) = a + bx$, through a set of data points, $y_i(x_i) = f(x_i)$. Chi-squared (X^2) is a function whose value is proportional to the summed squares of the difference between the actual y -value, $y_i(x_i)$, and the y -value of the calculated line, $y(x)$. In other words:

$$x^2 = f\left\{\sum[y_i - y]^2\right\}$$

Substituting the value of y from the linear solution $y(x) = a + bx$:

$$x^2 = f\left\{\sum[y_i - (a + bx)]^2\right\}$$

EXPONENTIAL FIT

The Exponential fit calculates an exponential curve through two cursor locations. The two cursors used in this calculation do not necessarily have to be on the same curve if the plot view displays a curve family. However, the cursors must not be positioned on the X-axis, and the cursors must not have the same x-value. Both of these situations will produce a zero denominator error in the calculation of the function's solution.



When configuring an exponential curve fit, the corresponding cursors must not be positioned on the X-axis and should not be positioned at the same X-value.

An exponential function is defined by the general equation:

$$y(x) = ae^{bx}$$

Consider two cursor locations at (x_1, y_1) and (x_2, y_2) . If both points lie on the same exponential curve, then:

$$y_1 = ae^{bx_1}, \text{ and}$$

$$y_2 = ae^{bx_2}.$$

Therefore, finding the exponential solution that satisfies both points is a problem of determining the values of a and b . Win4145 determines the values of a and b by determining the algebraic solution to the above system of two equations with two unknowns.

The physical meaning of a and b can be more easily visualized if the natural logarithm of the general equation is considered:

$$\ln[y(x)] = \ln(ae^{bx}) = \ln(a) + \ln(e^{bx}) = \ln a + bx$$

$$\ln[y(x)] = \ln a + bx$$

The above equation demonstrates that $\ln[y(x)]$ is the equation of a straight line. When the above equation is differentiated with respect to x , it is clear that the slope of $\ln[y(x)]$ is equal to b .

$$\frac{d}{dx} \ln[y(x)] = \frac{d}{dx} [\ln(a)] + \frac{d}{dx} (bx) = b$$

To understand the meaning of a , consider again the logarithmic form of the exponential solution at $x = 0$:

$$\ln[y(x)] = \ln(a) + bx$$

$$\ln[y(0)] = \ln(a)$$

$$y(0) = a$$

Therefore, if the exponential solution $y(x) = ae^{bx}$ is plotted on a log/lin plot, then the curve is a straight line, and it follows that:

a = y-intercept, and

b = slope.

The values of a and b are displayed in the respective Curve Fit Table.

TANGENT FIT

The Tangent fit calculates the equation of a line tangent to the data curve at a point designated by a single cursor. The Tangent fit is only functional when the Tracking option is active (refer to *Cursors Functions* earlier in this chapter). The Tangent fit cannot be selected if the associated cursor is positioned at the first or last data point of a sweep. The Tangent fit is also limited to cursor positions that are not located on a portion of the curve that is exactly vertical. This condition would cause a zero denominator error in the calculation of the function's solution. Win4145 will display the appropriate error message when either of these situations is detected.



The Tangent fit is only functional when the Tracking option is active. The Tangent fit cannot be selected if the associated cursor is positioned at the first or last data point of a sweep. The Tangent fit is also limited to cursor positions that are not located on a portion of the curve that is exactly vertical.

The general equation of the tangent line is:

$$y_{\text{tbl}}(x) = m_{\text{tbl}}x + d$$

The Tangent fit finds a solution by first calculating the quadratic equation defined by the cursor position and the bordering two data points: one on the left, and the other on the right. These three points are referred to as (x_1, y_1) , (x_2, y_2) , and (x_3, y_3) . (x_2, y_2) are the coordinates of the cursor. The quadratic equation is of the form:

$$f(x) = ax^2 + bx + c$$

The quadratic equation is interpolated using Neville's algorithm. The constants a, b, and c are expressed in terms of (x_1, y_1) , (x_2, y_2) , and (x_3, y_3) . The slope of the tangent line is calculated by evaluating the derivative of the quadratic equation at (x_2, y_2) :

$$m_{\text{tan}} = \frac{d}{dx}[f(x_2)] = 2ax_2 + b$$

The general equation of the tangent line now becomes:

$$y_{\text{tan}}(x) = [2ax_2 + b]x + d$$

The y-intercept (d) is found by evaluating the above equation at (x_2, y_2) :

$$d = y_2 - x_2(2ax_2 + b)$$

Therefore, the exact equation of the tangent line at (x_2, y_2) is:

$$y_{\text{tan}}(x) = (2ax_2 + b)(x - x_2) + y_2$$

where a, b, and c are functions of (x_1, y_1) , (x_2, y_2) , and (x_3, y_3) as defined by Neville's algorithm. The slope, X_{int} , and Y_{int} are reported in the respective Curve Fit Table.

GRADIENT FIT

The Gradient fit calculates a straight line through a cursor with the specified gradient. The cursor used in this calculation can be located anywhere in the plot window.

THE CURVE FIT TABLES

When one or more cursors are assigned to the plot window, the plot window will be resized to accommodate the Cursor Table and two Curve Fit Tables displayed underneath the plot area.

The two Curve Fit Tables display information about the respective fit routine configured in the plot window. The first row of the table displays the fit type: Cursor, Linear, Exponential, Tangent, or Gradient. If the Cursor, Linear, Tangent, or Gradient fit type is configured, the remaining rows of the curve fit table will report the slope, Y_{int} , and X_{int} of the fitted line. If the Exponential fit type is configured, the remaining rows will report the values of "A" and "B", the exponential constants. The parameters reported in the Curve Fit Tables are dynamic. This means that if any of the cursors are moved thus changing the fitted equation, the values displayed in the table are updated automatically.



When a plot window is printed in hard copy or copied to the Windows Clipboard, the Curve Fit Tables will be included in the result.



The Curve Fit Tables are displayed with the Cursor Table even if there are no curve fits configured in the plot window. If this is the case, a sequence of asterisks will occupy each row.

THE OPTIONS TOOLSET

The Options toolset includes the tools used to add information or edit the appearance of the plot window. The Options toolset is displayed or hidden using the last of the four vertical tool ribbon buttons located along the left border of the plot window. From left to right, the Options toolset consists of the following buttons:

1. **Legend ON/OFF:** Displays a line legend along the right boundary of the plot window.
2. **Test Conditions ON/OFF:** Displays the source unit setup conditions along the right boundary of the plot window.
3. **Plot Setup:** Displays the Setup Plot View dialogue box. See Creating/Editing Plot Window Definition for a full description of how to use the dialogue box.
4. **Titles:** Displays the Titles dialogue box. The Titles dialogue box provides the capability to add or edit the plot window title, axis labels, and a plot window comment line.
5. **Colors:** Displays the Colors dialogue box. The Colors dialogue box provides the capability to change the color of any plot window attribute.
6. **Zoom In:** Allows the user to specify a region of the active plot window on which to zoom.
7. **Zoom Out:** Zooms the active plot window back to it's original size and format.
8. **Overlay:** Allows the user to select up to four plot overlays to be displayed over the active plot window.
9. **Memory:** Allows the user to store an active plot window to memory to be utilized as an overlay plot.

LINE LEGEND

Selecting the Options toolset Legend button turns ON/OFF a line legend along the right boundary of the plot window.

Plot window curves can be distinguished in appearance by displaying each curve in a unique color. A line legend is most useful when each plot window curve is displayed in a separate color. This is done by editing the Plot Colors dialogue box described later in this section.



If the displayed legend does not include all of the data curves displayed in the plot window, the plot window is sized too small. Enlarging the plot window will correct the problem.

TEST CONDITIONS

Selecting the Options toolset Test Conditions button turns ON/OFF the source unit test conditions display along the right boundary of the plot window.

The Test Conditions button provides the capability to include the setup conditions of each active source unit as part of the plot window display. When the Test Conditions display is active, the output mode of each source unit will be identified along with any relevant output parameters. Below is a list of the output modes and the parameters reported with each mode. The list is not complete but includes the output functionality supported by most instrument source units. Signal type is distinguished by units (V or I).

CONSTANT OUTPUT

VALUE

STEP OUTPUT

Start
Stop
Step
Pnts

SWEEP OUTPUT

Start
Stop
Step
Pnts

For a comprehensive list and description of source unit output modes, refer to the chapter at the back of this manual that is specific to the instrument in question.

PLOT SETUP

Selecting the Options toolset Plot Setup button displays the Setup Plot View dialog box where basic plot setup attributes may be changed as described in earlier sections.

TITLES

The Options toolset Titles button is used to display the Plot Titles dialogue box. The Plot Titles dialogue box can also be displayed by selecting OPTIONS/TITLES from the plot mode menu bar. The Plot Titles dialogue box includes the fields necessary to add or edit the following plot window text:

1. Plot Title
2. X-Axis Label
3. Y₁-Axis Label
4. Y₂-Axis Label
5. Comment Line

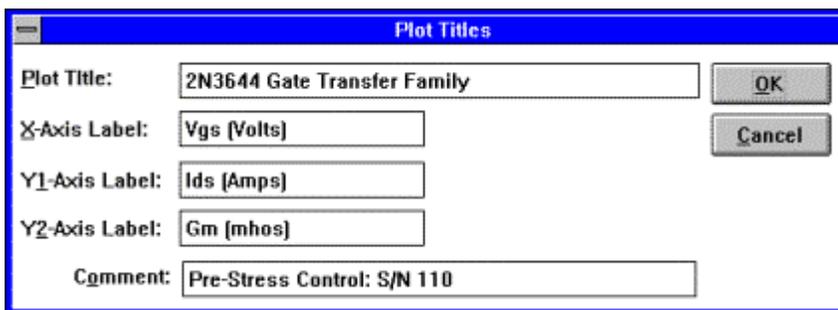


Figure 6-5: The Plot Titles Dialogue Box

COLORS

The Options toolset Colors button displays the Plot Colors dialogue box. The Plot Colors dialogue box provides the controls to edit the color composition of the active plot window. The default plot window colors can be edited from the corresponding controls in the Workspace dialogue box.

A plot window consists of seven (7) plot window objects. The color of each plot window object can be independently colored. The seven plot window objects are:

1. Data curves, comprised of markers and/or lines.
2. Axis numbers.
3. Plot title (Optional).
4. Cursors (Optional).
5. Axes (Y₂-Axis Optional).
6. Axis labels and a comment line (Optional).
7. Window background.

The user can designate the default color of each plot window object. Every time a plot window is created, the plot window will be displayed according to the default color specifications. The default plot color controls are located in the Workspace dialogue box. In addition to controlling the default appearance of each plot window, the user can customize the color composition of the active plot window without changing the appearance of the remaining plot windows in Win4145. The color composition of the active plot window is controlled from the Plot Colors dialogue box.

ZOOM IN

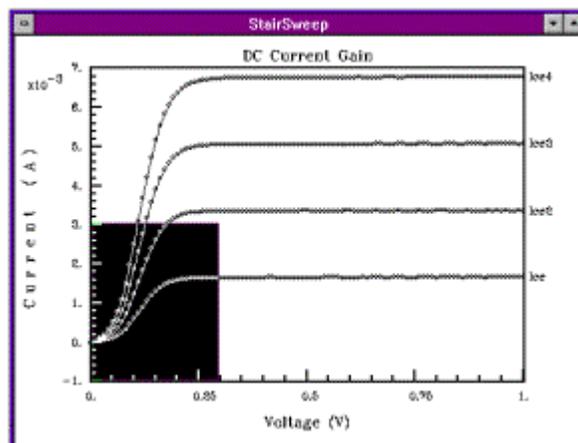
The zoom in feature provides the capability to update a plot window that is an enlarged view of the original plot window. The zoom feature is not a bitmap enlargement; instead, the zoom feature updates the plot window so that it is scaled according to the boundaries defined with the zoom cursor while in the original plot window.

ZOOM OUT

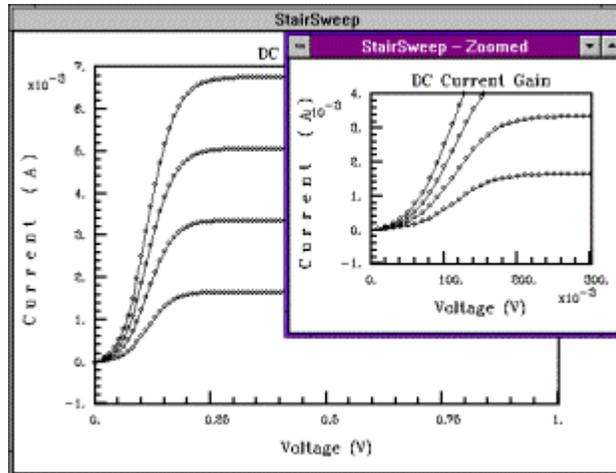
The zoom out feature provides the capability to return a plot window that has an enlarged view of the original plot window to its original scale.

HOW TO CREATE A PLOT WINDOW WITH THE ZOOM FEATURE:

1. Make certain that the plot in question is the active plot window. If it's not, click anywhere in the plot or click twice on the plot window icon.
2. Activate the zoom feature by clicking the toolbar Zoom button. When the pointer is returned to the active plot window, the cursor will turn into a set of crosshairs.
3. Select a location in the plot window that will be one of the corners of the zoomed view. Click and hold the left mouse button. While holding the left button, drag the cursor to designate the boundary of the new plot.
4. Release the left mouse button to display the updated plot window. The boundaries of the updated plot window will be limited to the area designated with the crosshairs in the original view.



After clicking on the tool bar ZOOM button, move the Zoom Cursor to a position on the plot window that will be the corner reference of the new plot. Click and drag the mouse in order to designate the boundaries of the zoomed view.



The updated plot window will be displayed after the mouse button is released. In this view, the updated plot window is an enlargement of the 4mA by 0.3V area of the original plot window.

OVERLAY

The overlay feature allows the user to select up to four plots to overlay upon the active plot window



The plot that is being overlaid MUST be the same scale as the plot that is the overlay. The plots are not automatically rescaled.

HOW TO OVERLAY PLOTS:

1. Make certain that the plot to be overlaid upon is the active plot window. If it's not, click anywhere in the plot or click twice on the plot window icon.
2. Activate the overlay feature by clicking the toolbar OVERLAY button. Then from the four overlay buttons select the desired overlay plot by clicking on the corresponding button. Overlay 1 corresponds to a overlay plot stored in memory 1 and so on. *Refer to the Memory feature below for a description on how to use the memory feature.*

MEMORY

The memory feature allows the user to store up to four plots in memory for use in the overlay feature.



The plot that is being overlaid **MUST** be the same scale as the plot that is the overlay. The plots are not automatically rescaled.

HOW TO STORE PLOTS FOR OVERLAYING:

1. Make certain that the plot to be stored for overlay is the active plot window. If it's not, click anywhere in the plot or click twice on the plot window icon.
2. Activate the memory feature by clicking the toolbar MEMORY button. Then from the four memory buttons select the desired memory location for storage of the overlay plot by clicking on the corresponding button.
Memory 1

CHAPTER 7: PRINTING, EXPORTING, AND LINKING

PRINTING DATA AND PLOT WINDOW CONTENTS

Win4145 supports a full range of printing, exporting, and database searching/exporting options. Data window spreadsheets and plot windows can be printed or copied to the Windows clipboard with the click of a single toolbar button. Project files can be exported and imported allowing data collected on one machine to be shared with another.

PRINTING DATA AND PLOT WINDOWS



To print a data or plot window, click the tool bar Print button.

HOW TO PRINT A DATA WINDOW SPREADSHEET OR PLOT WINDOW:

Make certain that the desired data window spreadsheet or plot window is the active desktop window. If not, click anywhere in the window or double click on the window icon

1. In the case of the data window spreadsheet highlight the desired data to be printed.
2. Click the tool bar Print button.

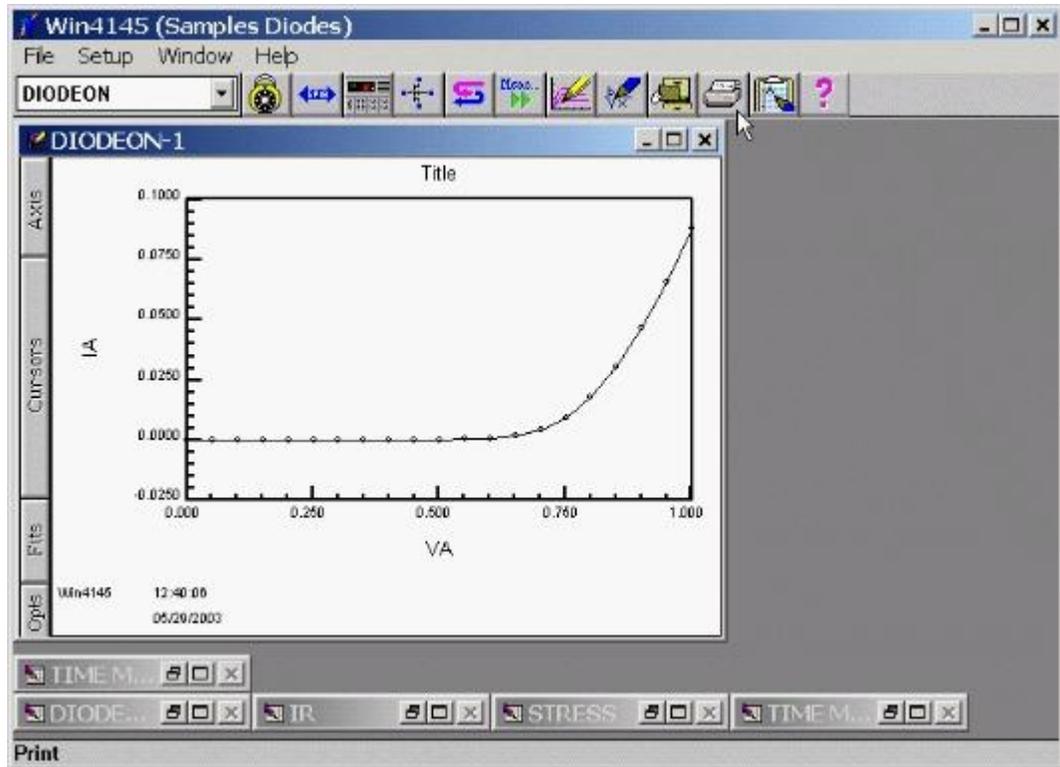


Figure 7-1: How to Print a Data Window Spreadsheet or a Plot Window.

COPYING A DATA OR PLOT WINDOW TO THE CLIPBOARD

A data window spreadsheet or plot window can be copied to the Windows Clipboard and pasted into any application supporting the clipboard.



To copy a data or plot window to the Windows Clipboard, click the tool bar Clipboard button.

HOW TO COPY A DATA OR PLOT WINDOW TO THE CLIPBOARD

1. Activate the desired data window spreadsheet or plot window.
2. In the case of the data window spreadsheet highlight the desired data to be copied to the clipboard.
3. Click the tool bar Clipboard button.

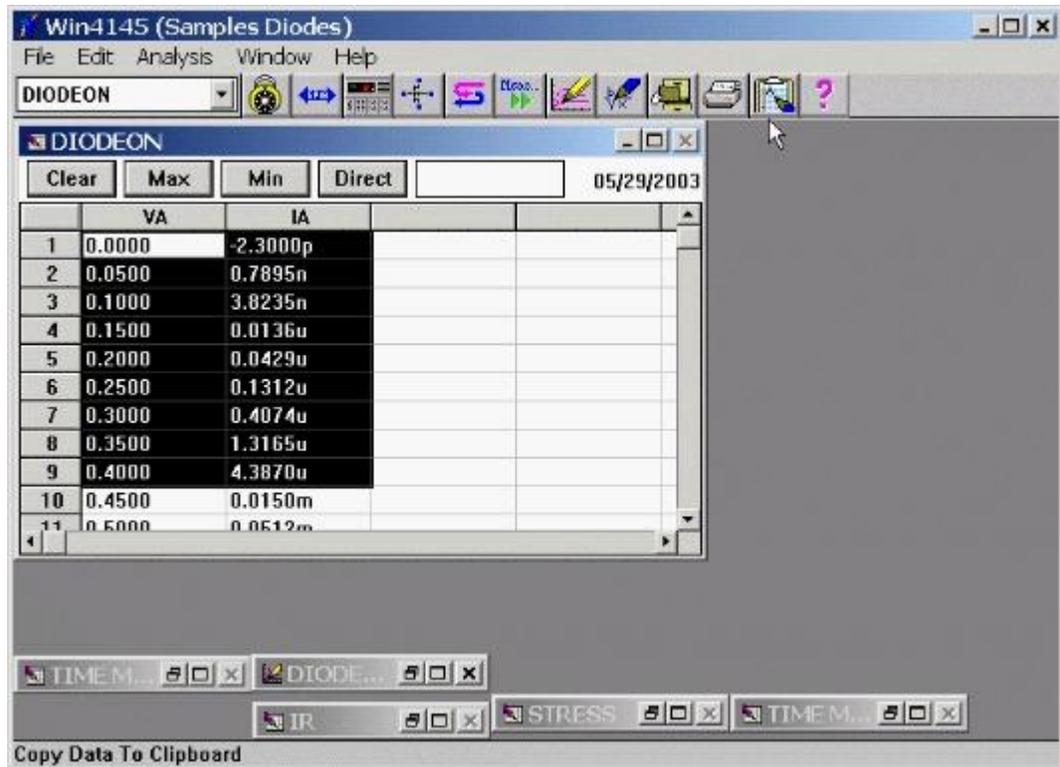


Figure 7-2: How to Copy a Data or Plot Window to the Clipboard.

SHARING DATA BETWEEN WIN4145 WORKSTATIONS

Win4145 allows the sharing of data between Win4145 workstations through binary exporting and importing of project data.

Project File Binary Export

Project files exported in binary format store all of the information included in the original project file. This includes all of the associated test setups, the instrument driver selection(s) required to execute the test setups, and all of the measured data and plot windows corresponding to the test setups. Project files exported in binary format can be imported by a second Win4145 workstation. Because the binary files include all of the information stored by the original project file, imported binary files can be executed and saved as if they were created on the destination machine. By exporting and importing project files, executable test setups and data can be shared between users and workstations. The purpose of the Binary Export function is pictured in Figure 7-3.

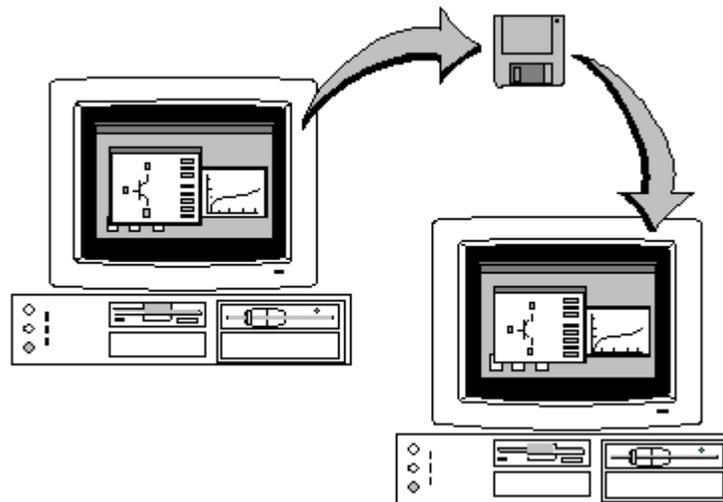


Figure 7-3: The Binary Export Function Provides the Capability to Copy Project Files from one Machine to Another.

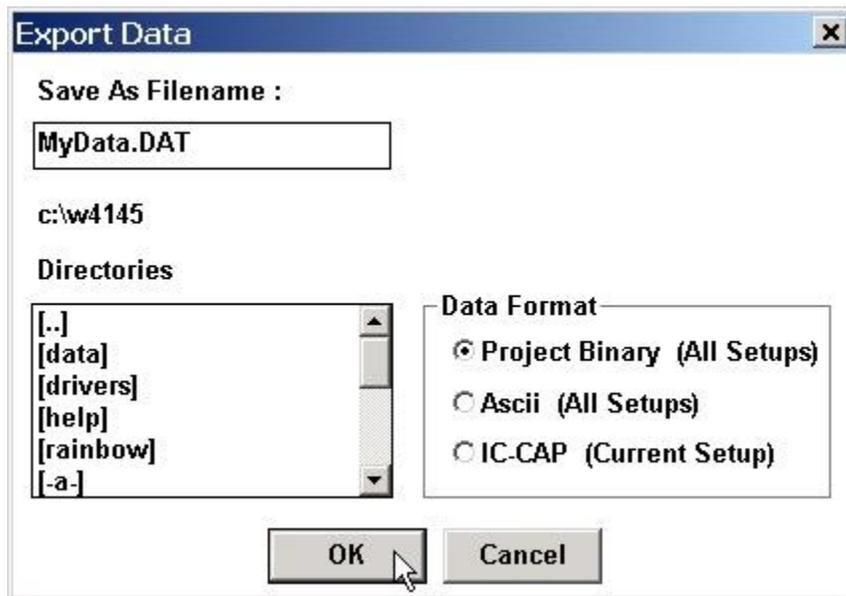
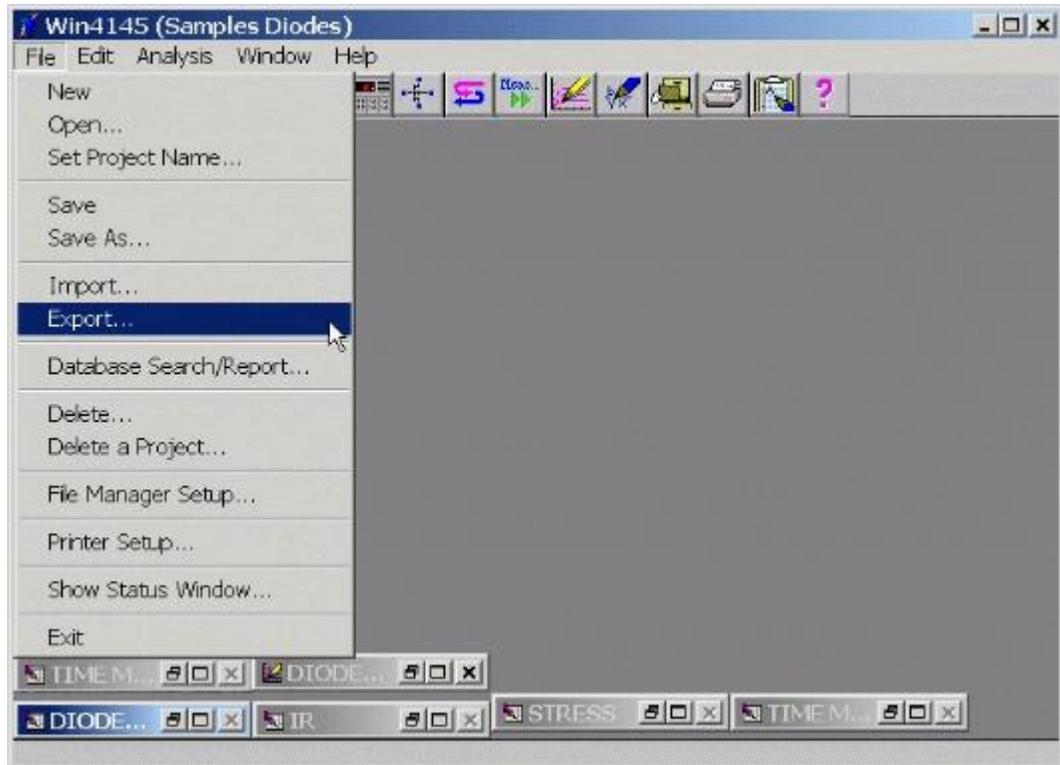


Figure 7-4: How to Export a Project File in Binary Format

HOW TO EXPORT A PROJECT FILE IN BINARY FORMAT

1. The active project file is the project file that will be exported when the Binary Export function is selected. If necessary, open the desired project file from the File Manager dialogue box.
2. Select FILE/EXPORT from any of the three menu bars. This will open the Export Data dialogue box.
3. Designate the destination path in the Directory window. Use the scroll arrows to move up and down the list box. Double click to make a selection. The designated path will appear above the Directory window. Depending on the size of the project file, it may be faster exporting to a hard-drive location and then using DOS-copy to move the file(s) to a diskette.
4. In the Save As Filename field, designate a DOS filename for the file you are exporting.
5. Select the Binary switch. This is important. If you export in ASCII format, the project file cannot be imported by another machine.
6. Click OK. The Export Data dialogue box will close automatically after Win4145 has completed exporting the file.

Project File Binary Import

Only project files that have been exporting in binary format can be imported to the desktop. There is no distinction between an imported project file and a project file opened from the File Manager. Imported project files include all of the test setup information, driver connections, and data results stored in the original project file. All of the test setups corresponding to the imported project file can be executed and edited just like the test setups created with the host machine. To save the imported file permanently, select FILE/SAVE AS from any menu bar.

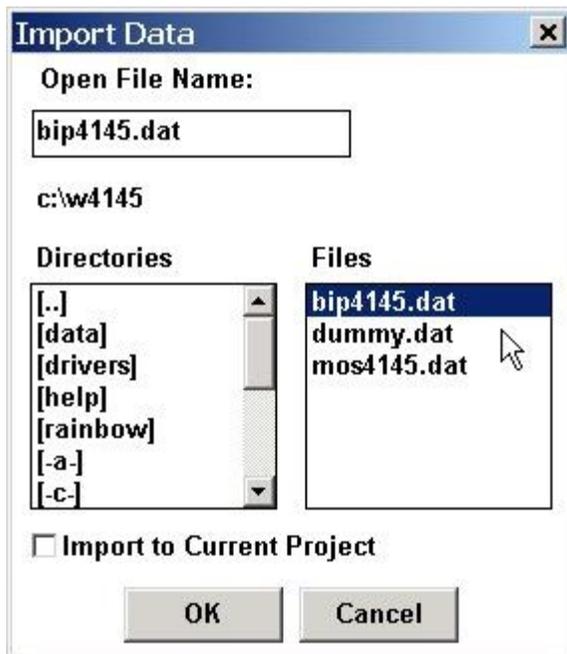
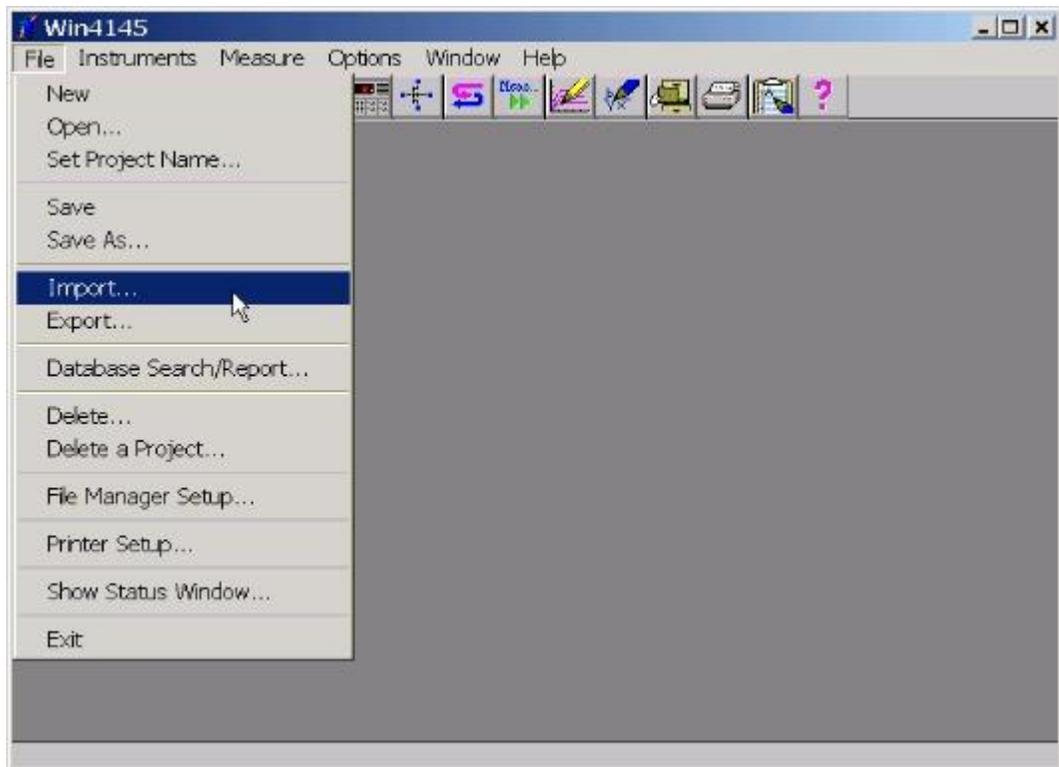


Figure 7-5: How to Import a Project File

HOW TO IMPORT A PROJECT FILE:

1. Select FILE/IMPORT from any of the three menu bars. This will open the Import Data dialogue box.
2. In the Directory window, designate the directory from which the project file will be imported. Use the scroll box and scroll arrows to move up and down the list. Double click to make a selection. The selected directory will appear above the Directory window.
3. In the Files window, designate the name of the project file that will be imported. The filename will appear in the Open File Name field. Double clicking on the desired filename will initiate the import function.
4. The import function can also be initiated by clicking once on the filename and clicking the OK button. The Import Data dialogue box will close automatically after the import function is complete.

EXTRACTING ASCII DATA FROM WIN4145

Files exported as ASCII data are readable text files only and do not include all of the information written in a binary version of the same file. Exported project files of this type *cannot* be imported and executed by another machine. An ASCII Export file includes a project file header, a header for each test setup, test setup conditions, data, and test results corresponding to each test setup included in the ASCII Export file. All setups contained in the source project file will be exported. The data which is exported for a particular setup may be controlled by defining a data view which allows the user to select which data vectors will appear in the ASCII export file.

HOW TO EXPORT PROJECT FILE DATA IN ASCII FORMAT

1. Open the desired project file from the File Manager dialogue box. If necessary, measure the desired test setups.
2. Select FILE/EXPORT from any of the three menu bars. This will open the Export Data dialogue box.
3. Designate the destination path in the Directory window. Use the scroll arrows to move up and down the list box. Double click to make a selection. The designated path will appear above the Directory window.
4. In the Save As Filename field, designate a DOS-filename for the file you are exporting.
5. Select the ASCII switch. This is important. If you export in binary format, you will not be able to read the file in an ASCII text editor.
6. Click OK. The Export Data dialogue box will close automatically after Win4145 has completed exporting the file.

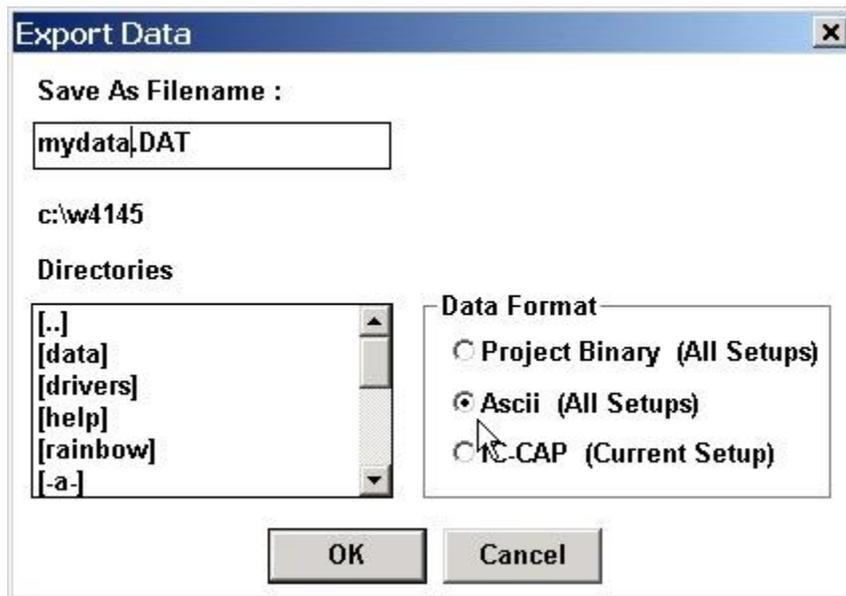
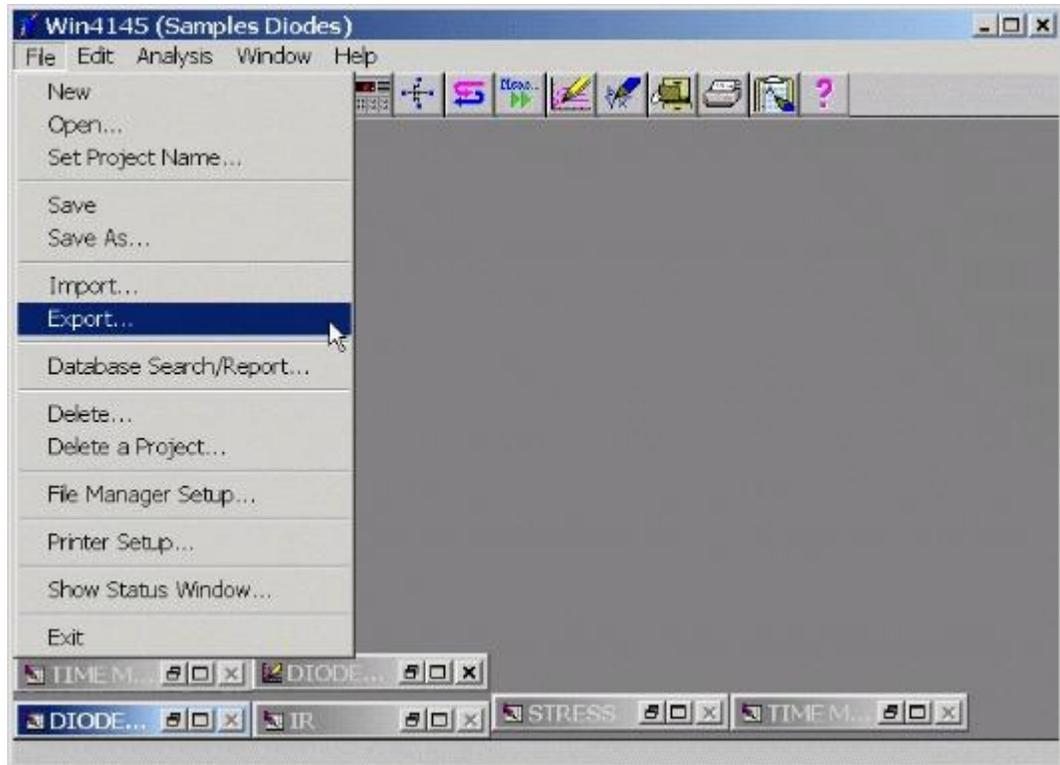


Figure 7-6: How to Export a Project File in ASCII Format